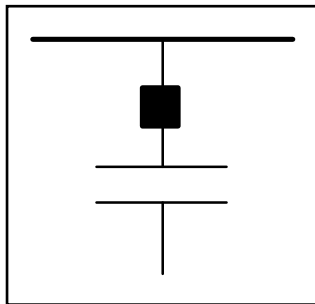
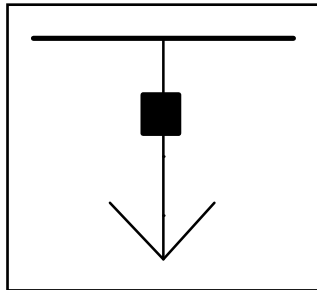
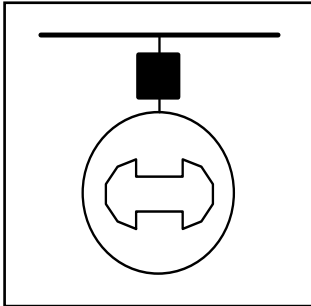
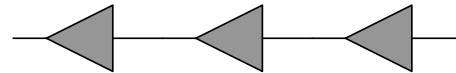


***PowerWorld***  
Corporation



## Simulator Version 13



Interactive power system  
simulation, analysis and  
visualization

## User's Guide

2001 South First Street  
Champaign, IL 61820  
<http://www.powerworld.com>  
[support@powerworld.com](mailto:support@powerworld.com)  
(217) 384-6330



# Table of Contents

<b>Chapter 1: Getting Started With PowerWorld.....</b>	<b>1</b>
About this Manual .....	2
Introduction to PowerWorld Simulator .....	3
Introduction to Simulator Add-On Tools .....	4
What's New.....	5
Help .....	9
Contact Information.....	10
Windows Basics .....	11
PowerWorld Simulator: Getting Started.....	12
Edit Mode Introduction.....	13
Run Mode Introduction.....	14
Script Mode Introduction .....	15
Message Log .....	16
Custom and Memo Display .....	17
Status Bar .....	18
Ribbons User Interface.....	19
Quick Access Toolbar .....	22
Add Ons Ribbon Tab .....	23
Case Information Ribbon Tab.....	26
Draw Ribbon Tab.....	29
Clipboard Ribbon Group.....	31
Formatting Ribbon Group .....	33
Individual Insert Ribbon Group .....	35
Quick Insert Ribbon Group.....	37
Select Ribbon Group .....	39
Onelines Ribbon Tab.....	40
Active Ribbon Group .....	42
Zoom Ribbon Group.....	46
Options Ribbon Tab .....	47
Tools Ribbon Tab.....	49
Edit Mode Ribbon Group.....	51
Other Tools Ribbon Group .....	52
Run Mode Ribbon Group.....	54
Power Flow Tools Ribbon Group.....	56
Window Ribbon Tab.....	59
<b>Chapter 2: Using Online Diagrams .....</b>	<b>62</b>
Online Diagram Overview .....	63

Relationship Between Display Objects and the Power System Model.....	64
Online Local Menu.....	65
Online Display Options Dialog.....	67
Display Options .....	68
Pie Chart/Gauge Options .....	70
Animated Flows Options .....	72
Thumbnail View .....	75
Thumb Nail Menu .....	76
Display Object Options .....	77
Substation Display Options .....	79
Online Animation .....	81
Copying Onelines to Other Programs .....	82
Display Explorer.....	83
Supplemental Data .....	84
Supplemental Data Fields on Onelines .....	86
Movie Maker.....	87
Map Projections.....	88
Great Circle Distance Dialog.....	92
Great Circle Distance Calculation .....	93
Populate Lon,Lat with Display X,Y .....	94
Geography/Coordinates .....	95
Shape File Import .....	96
GIS Shapefile Data: Control .....	97
GIS Shapefile Data: Identify .....	98
GIS Shapefile Data: Modify .....	99
GIS Shapefile Data: Format .....	100
GIS Shapefile Data: Shape List.....	101
Shapefile Database Record Dialog .....	102
Export Online As Shapefile .....	103
Insert Measure Line .....	105
Delete All Measure Lines.....	106
Path Distances from Bus or Group .....	107
Closet Facilities to Point.....	108
Online Screen Coordinates .....	109
Online Zooming and Panning .....	110
Save View Level Dialog.....	112
Window Ribbon Tab .....	114
Online Conditional Display of Objects.....	117
Keyboard Short Cut Actions Dialog .....	118
Printing Online Diagrams .....	120



Print Options Dialog .....	121
Printer Setup .....	123
<b>Chapter 3: Creating, Loading, and Saving Simulator Cases .....</b>	<b>124</b>
Application File Menu .....	125
Case Formats .....	126
GE EPC File Load Options .....	128
Validating a Case .....	129
Opening a Simulation Case .....	130
Opening a Oneline Diagram .....	131
Recently Opened Cases .....	132
Building a New Case .....	133
Building a New Oneline .....	134
Saving Cases .....	135
Saving a Oneline.....	136
Close Oneline .....	137
Auxiliary File Format (*.aux) .....	138
Generator Capability Curves Format (*.gcp) .....	139
Generator Cost Data Format (*.gcd).....	140
Generator Cost Curve Data Options .....	141
Identify Cross Compound Units .....	142
Injection Groups Format (*.inj) .....	143
Interface Data Format (*.inf).....	144
Sequence Data Format.....	145
Exporting Onelines in Different Graphic Formats .....	147
Saving Images as Jpegs .....	148
Saving Admittance Matrix and Jacobian Information .....	149
<b>Chapter 4: Building a Oneline Diagram .....</b>	<b>150</b>
Edit Mode Overview .....	151
Edit Mode General Procedures .....	152
Draw Ribbon Tab.....	154
Oneline Display Options.....	156
Anchored Objects .....	157
Inserting and Placing Multiple Display Fields.....	158
Area Display Objects .....	159
Area Display Options Dialog.....	160
Zone Display Objects .....	161
Zone Display Options Dialog .....	162
Super Area Display Objects.....	163
Super Area Display Options Dialog .....	164
Owner Display Objects .....	165

Owner Display Options Dialog .....	166
Area Fields on Onelines .....	167
Zone Fields on Onelines .....	168
Super Area Fields on Onelines.....	170
Owner Fields on Onelines .....	171
Bus Display Objects .....	172
Bus Fields on Onelines .....	173
Old Voltage Gauges .....	174
Old Voltage Gauge Options Dialog .....	175
Substation Display Objects .....	176
Substation Fields on Onelines .....	177
Generator Display Objects .....	178
Generator Fields on Onelines .....	179
Load Display Objects.....	180
Load Fields on Onelines.....	181
Transmission Line Display Objects .....	182
Line Fields on Onelines.....	183
Circuit Breakers on Onelines .....	184
Line Flow Pie Charts on Onelines.....	185
Line Flow Gauges.....	186
Line Flow Gauge Options Dialog .....	187
DC Transmission Line Display Objects.....	188
Multi-section Line Display Objects .....	189
Transformer Display Objects.....	190
Transformer Fields on Onelines.....	192
Series Capacitor Display Objects .....	193
Series Capacitor Fields on Onelines .....	194
Switched Shunt Display Objects .....	195
Switched Shunt Fields on Onelines .....	196
Interface Display Objects.....	197
Automatically Inserting Interface Display Objects .....	198
Interface Fields on Onelines.....	199
InterArea Flow Options Dialog .....	200
Interface Pie Charts on Onelines .....	201
Loading NERC Flowgates .....	202
Saving NERC Flowgates.....	203
Links to Onelines and Auxiliary Files .....	204
Document Links on Onelines.....	206
Background Lines on Onelines .....	207
Background Rectangles on Onelines .....	208

Background Ellipses on Onelines .....	209
Background Pictures on Onelines.....	210
Background Lines Dialog .....	211
Converting Background Lines.....	213
Converting Background Ellipses.....	214
Converting Background Rectangles.....	215
Online Fields.....	216
Text on Onelines .....	217
Memo Text.....	218
Generic Model Fields .....	219
Pie Charts/Gauges: Lines.....	220
Pie Charts/Gauges: Interfaces.....	222
Pie Charts/Gauges: Pie Chart/Gauge Styles .....	223
Pie Charts/Gauges: General Options .....	224
Pie Chart / Gauge Dialogs.....	225
Pie Chart / Gauge Style Dialog .....	229
Pie Chart / Gauge Style Dialog - Standard Parameters Tab .....	231
Pie Chart / Gauge Style Dialog - Open Parameters Tab .....	233
Pie Chart / Gauge Style Dialog - Pie Chart Parameters Tab .....	234
Pie Chart / Gauge Style Dialog - Gauge Parameters Tab.....	235
Pie Chart / Gauge Example .....	237
<b>Chapter 5: Edit Mode Tools and Options .....</b>	<b>238</b>
Appending a Case.....	239
Refresh Anchors .....	240
Bus Renumbering Dialog .....	241
Bus Renumbering: Automatic Setup of Bus List Options .....	242
Bus Renumbering: Bus Change Options .....	244
Renumber Areas/Zones/Substations Dialog .....	245
Merging Buses.....	246
Splitting Buses.....	247
Split Bus Dialog .....	248
Equipment Mover.....	249
Potential Misplacements Dialog .....	250
Tapping Transmission Lines .....	251
Automatic Line Tap Dialog .....	252
Setting Default Drawing Options .....	254
Bus Selection Page.....	257
List Unlinked Display Objects .....	259
Set Selected Field .....	261
Network Cut.....	262

Browse Open Onelines .....	264
Unused Bus Numbers.....	265
Palette Overview.....	266
Using the Insert Palettes .....	267
Automatically Inserting Buses .....	270
Automatically Inserting Transmission Lines .....	272
Automatically Inserting Generators.....	274
Automatically Inserting Loads .....	275
Automatically Inserting Switched Shunts.....	276
Automatically Inserting Interface Display Objects .....	277
Automatically Inserting Substations .....	278
Automatically Inserting Borders.....	279
Equivalents .....	282
Equivalents Display .....	283
Scaling.....	286
Overview of Facility Analysis in PowerWorld Simulator.....	290
Facility Analysis Dialog .....	291
Augmenting Path Max Flow Min Cut Algorithm.....	293
Graph Flow .....	294
<b>Chapter 6: Editing Online Diagrams .....</b>	<b>295</b>
Clipboard Ribbon Group.....	296
Select by Criteria Dialog .....	298
Select Ribbon Group .....	300
Grid/Highlight Unlinked Objects.....	301
Using the Online Alignment Grid .....	302
Setting Background Color .....	303
Zoom, Pan, and Find .....	304
Default Drawing Values .....	306
Formatting Ribbon Group .....	307
Format Multiple Objects.....	309
Paste Format Dialog.....	310
Font Properties .....	311
Line/Fill Properties .....	312
Levels/Layers Options .....	313
Screen Layers.....	314
Screen Layer Options .....	315
Format Field Properties .....	316
Display/Size Properties .....	317
Align Group Objects .....	318
Clipboard Ribbon Group.....	319

Undo Command.....	321
<b>Chapter 7: Properties of Simulator Objects .....</b>	<b>322</b>
Zone Information (Edit Mode).....	323
Bus Information (Edit Mode).....	325
Bus Field Information .....	328
Shortest Path Between Elements.....	330
Substation Information (Edit Mode).....	331
Substation Field Options.....	333
Generator Information (Edit Mode).....	335
Generator Field Information.....	337
Generator Options: Display.....	339
Generator Options: Power and Voltage Control.....	340
Generator Options: Costs .....	342
Generator Options: Fault Parameters.....	344
Generator Options: Owners, Area, Zone, Sub .....	345
Generator Cost Description .....	346
Set Generator Participation Factors.....	347
Generator Reactive Power Capability Curve .....	349
Load Information (Edit Mode) .....	350
Load Options: Load Information.....	352
Load Options: OPF Load Dispatch.....	353
Load Field Information .....	354
Load Modeling .....	356
Line/Transformer Information (Edit Mode) .....	357
Transmission Line/Transformer Options: Display.....	359
Transmission Line/Transformer Options: Parameters.....	360
Line Shunts Information .....	361
Line Per Unit Impedance Calculator Dialog .....	362
Transmission Line Parameter Calculator .....	363
Transmission Line/Transformer Options: Transformer Control.....	373
Transformer AVR Dialog .....	375
Transmission Line/Transformer Options: Series Capacitor.....	377
Multi-Section Line Information .....	378
Line Field Options .....	380
Series Capacitor Field Options Dialog .....	382
Transformer Field Options Dialog .....	383
DC Transmission Line Options.....	385
DC Line Options: Line Parameters .....	386
DC Line Options: Rectifier Parameters .....	388
DC Line Options: Inverter Parameters.....	389

DC Line Options: Actual Flow .....	390
DC Line Options: OPF.....	391
Multi-Terminal DC Record Information .....	392
Multi-Terminal DC Bus Information.....	393
Multi-Terminal DC Converter Information .....	394
Multi-Terminal DC Line Information .....	396
Transmission Line/Transformer Options: Transformer Control.....	397
Transformers Bases and Impedances Dialog .....	399
Transformer Impedance Correction Table Display .....	400
Transformer AVR Dialog .....	401
Transformer Mvar Control Dialog.....	403
Transformer Phase Shifting Information.....	405
Transformer Field Options Dialog .....	407
Three Winding Transformer Information .....	409
Switched Shunt Information (Edit Mode).....	411
Switched Shunt Field Information.....	416
Bus Information (Run Mode).....	418
Substation Information (Run Mode).....	421
Generator Information (Run Mode) .....	423
Generator Information: Power and Voltage Control .....	425
Generator Options: Costs .....	427
Generator Information: OPF .....	429
Generator Options: Owners, Area, Zone, Sub .....	430
Load Information (Run Mode).....	431
Line/Transformer Information (Run Mode) .....	434
Transformer AVR Dialog .....	438
Transformer Mvar Control Dialog.....	440
Transformer Phase Shifting Information.....	442
Transformer Impedance Correction Tables Dialog.....	444
Switched Shunt Information (Run Mode) .....	445
Zone Information (Run Mode).....	449
Labels.....	451
Label Manager Dialog.....	455
Area Information .....	457
Area Information: Buses.....	459
Area Information: Gens .....	460
Area Information: Loads.....	461
Area Information: Area MW Control Options.....	462
Area Information: Info/Interchange.....	463
Area Information: Options .....	464

Area Information: Tie Lines .....	465
Area Information: OPF .....	466
Area Information: Custom .....	467
Area Field Options .....	468
Super Area Information Dialog .....	471
Super Area Field Information .....	474
Interface Information.....	476
Interface Element Information .....	478
Interface Field Information Dialog .....	479
Interface Pie Chart Information Dialog.....	480
Automatically Inserting Interfaces in Case.....	481
Nomogram Information Dialog .....	482
<b>Chapter 8: Text-based Information Displays .....</b>	<b>484</b>
Model Explorer .....	485
Case Information Displays .....	488
Configuring the Case Information Displays .....	489
Custom Field Descriptions.....	492
Custom Field Description Dialog .....	494
Calculated Fields.....	496
Case Information Customizations Display .....	498
Case Information Displays: Local Menu Options.....	499
Case Information Displays: Colors and Cell Styles .....	501
Case Information Displays: Using Cell Handles .....	502
Case Information Displays: Sorting Records .....	503
Case Information Displays: Finding Records .....	504
Saving Case Information Display Contents as HTML Tables.....	505
HTML Table Format Dialog .....	506
Case Information Toolbar .....	507
Case Information Toolbar: Copy, Paste and Send Menu .....	509
Case Information Toolbar: Filtering Menu .....	511
Case Information Toolbar: Geo Data View Menu.....	513
Case Information Toolbar: Load Auxiliary Files Menu.....	514
Case Information Toolbar: Save Auxiliary Files Menu.....	515
Case Information Toolbar: Records Menu.....	517
Case Information Toolbar: Set, Toggle, and Columns Menus .....	519
Area/Zone/Owner Filters .....	521
Advanced Filtering .....	522
Advanced Filters Dialog .....	523
Advanced Filters Display.....	526
Custom Expressions .....	527

Custom Expressions Display .....	528
Functions and Operators Available.....	529
Find Dialog Basics.....	532
Search for Text Dialog.....	534
Model Conditions Display and Dialog.....	535
Model Filters Display and Dialog .....	536
Entering a Range of Numbers .....	537
Copying Simulator Data to and from Other Applications .....	538
Save Case Information Data .....	539
Key Fields .....	540
Required Fields.....	542
Contour Column Dialog .....	543
Contour Column Type .....	544
Grid Metrics Dialog.....	546
Auxiliary File Export Format Description (For both Display and Power System) ...	547
Geographic Data View .....	550
Geographic Data View Styles: Fields and Attributes.....	552
Geographic Data View Styles: General Display Options.....	556
Geographic Data View Styles .....	557
Custom Case Information Display .....	559
Define Fields/Strings .....	560
Change Field Data .....	561
Show Fields Primary.....	562
Show Fields Secondary.....	564
Custom Case Information Display Local Menu.....	565
Case Description .....	566
Case Summary .....	567
Power Flow List .....	569
Quick Power Flow List .....	571
Outages .....	572
Area Display .....	573
Zone Display .....	575
Tie Lines between Areas Display .....	576
Tie Lines between Zones Display .....	578
Super Area Display.....	579
Bus Display .....	581
Remotely Regulated Bus Display .....	582
Bus Mismatches Display .....	584
Substation Records Display .....	585
Generator Display .....	586



Generator/Load Cost Models .....	588
Generator Cost Models Display .....	589
Generator Cubic Cost Display .....	591
Generator Piecewise Linear Cost Display .....	593
Load Display.....	595
Load Benefit Models Display .....	596
Line and Transformer Display .....	598
Multi-Section Lines Display .....	600
Transformer Display.....	602
Three Winding Transformer Display.....	604
Line Shunts Display.....	605
DC Lines Display .....	606
Multi-Terminal DC Record Display.....	607
Switched Shunt Display .....	608
Interface Display .....	610
Nomogram Display.....	612
Injection Group Display .....	613
Injection Group Dialog .....	615
Participation Point Records Display .....	617
Island Display.....	620
MW Transactions Display.....	621
Transaction Dialog .....	622
Limit Monitoring Settings .....	624
Limit Monitoring Settings and Limit Violations Dialog.....	625
Limit Group Dialog .....	629
Owner Data Information Display .....	632
Owner Dialog .....	633
Owned Bus Records Display.....	634
Owned Load Records Display .....	635
Owned Generator Records Display.....	636
Owned Line Records Display .....	637
Jacobian Display .....	638
Ybus Display.....	639
<b>Chapter 9: Injection Groups.....</b>	<b>640</b>
Injection Group Overview .....	641
Creating Injection Groups.....	642
Auto Insert Injection Groups.....	643
Deleting Injection Groups .....	645
Injection Group Display .....	646
Injection Group Dialog .....	648

Import PTI Subsystems Dialog .....	650
Participation Points Overview .....	651
Participation Point Records Display .....	652
Add Participation Points Dialog .....	655
<b>Chapter 10: Solving and Simulating a Case .....</b>	<b>658</b>
Simulator Options .....	659
Power Flow Solution Options.....	660
Power Flow Solution: Common Options.....	661
Power Flow Solution: Advanced Options .....	664
Post Power Flow Solution Actions Dialog .....	667
Power Flow Solution: Island-Based AGC.....	669
Power Flow Solution: DC Options .....	671
DC Power Flow Loss Setup .....	673
Power Flow Solution: General .....	674
Power Flow Solution: Storage .....	675
Message Log Options .....	676
Environment Options.....	677
Online Options.....	679
Case Information Display Options.....	682
Limits Options .....	685
File Management Options .....	686
Power Flow Tools Ribbon Group.....	688
Solving the Power Flow .....	691
Area Control.....	692
Set Generator Participation Factors.....	695
Transaction Dialog .....	697
Calculate MW-Distance.....	699
MW-Distance Options.....	700
Area Control Error (ACE) Chart.....	701
Area Load and Generation Chart .....	702
Area Losses Chart.....	703
Area MW Transactions Chart.....	704
Area Average Cost Chart .....	705
Bus View Display .....	706
Substation View Display .....	709
<b>Chapter 11: Run Mode Tools and Options .....</b>	<b>710</b>
Generator Economic Curves.....	711
Find Branches that Create Islands .....	712
Custom Hint Values.....	713
Line Loading Replicator .....	715

Dynamic Formatting Overview .....	718
Dynamic Formatting Dialog.....	719
Difference Flows .....	721
Difference Flows: Case Types .....	722
Difference Flows Dialog .....	723
Using Difference Flows .....	724
Present Topological Differences from Base Case .....	726
Governor Power Flow .....	728
Governor Power Flow: Generator Options (Ignore Area/Zone/Owner filter) Tab..	729
Governor Power Flow: Options Tab .....	730
Contingency Analysis: An Introduction .....	731
Available Contingency Actions.....	732
Contingency Analysis: Terms .....	733
Contingency Solution Options Dialog .....	734
Contingency Case References .....	735
Contingency Case References - State Information.....	736
Contingency Case References - Defining the Reference State.....	738
Contingency Case References - Reference State Solution Options.....	739
Contingency Analysis: Defining Contingencies .....	740
Auto Insert Contingencies.....	741
Loading Contingencies from a File .....	744
PSS/E Contingency Format .....	745
PSS/E Load Throw Over Files .....	746
Saving Contingency Records to a File.....	747
Global Actions .....	748
Contingency Blocks .....	749
Contingency Analysis Dialog Overview .....	752
Running the Contingency Analysis .....	753
Other Contingency Actions.....	754
Contingency Analysis Dialog - Contingencies Tab.....	757
Contingency Violations Display .....	760
Contingency Definition Display .....	762
Lines, Buses and Interfaces Tab.....	764
Lines, Buses and Interfaces Tab - Contingencies Section .....	765
Contingency Definition Display .....	766
Contingency Options Tab.....	768
Contingency Options Tab: Modeling.....	769
Contingency Options Tab: Limit Monitoring .....	771
Contingency Options Tab: Advanced Modeling.....	774
Model Expressions .....	777

Bus Load Throw Over Records .....	782
Contingency Options Tab: Report Writing .....	783
Contingency Options Tab: Miscellaneous.....	785
Contingency Summary Tab .....	786
Comparing Contingency Analysis Results.....	787
Comparing Contingencies List Displays .....	789
Contingency Definition Dialog .....	792
Contingency Element Dialog .....	794
Make-Up Power Sources.....	805
Contingency Make-Up Sources Dialog .....	806
Tutorial: Contingency Analysis .....	807
Tutorial: Contingency Analysis - Page 2 .....	808
Tutorial: Contingency Analysis - Page 3 .....	811
Tutorial: Contingency Analysis - Page 4 .....	813
Tutorial: Contingency Analysis - Page 5 .....	815
Tutorial: Contingency Analysis - Page 6 .....	816
Tutorial: Contingency Analysis - Page 7 .....	818
Tutorial: Contingency Analysis - Page 8 .....	819
Tutorial: Contingency Analysis - Page 9 .....	820
Tutorial: Contingency Analysis - Page 10.....	821
Tutorial: Contingency Analysis - Page 11.....	822
Fault Analysis .....	823
Fault Analysis Dialog .....	824
Fault Analysis Bus Records .....	827
Fault Analysis Generator Records .....	828
Fault Analysis Line Records.....	829
Mutual Impedance Records.....	830
Mutual Impedance Record Dialog .....	831
Fault Analysis Load Records.....	832
Fault Analysis Switched Shunt Records .....	833
Contouring .....	834
Contouring Options .....	835
Contour Type .....	836
Contour Type Options .....	840
Custom Color Map .....	842
Functional Description of Contour Options.....	844
Power Transfer Distribution Factors.....	848
Power Transfer Distribution Factors Dialog .....	849
Line Outage Distribution Factors (LODFs).....	853
Line Outage Distribution Factors Dialog .....	855

Advanced LODF Calculation Dialog .....	857
Directions Display .....	858
Directions Dialog.....	859
Auto Insert Directions .....	860
Flow and Voltage Sensitivities .....	861
Loss Sensitivities.....	863
Transmission Loading Relief Sensitivities.....	866
TLR Sensitivities Dialog .....	867
TLR Multiple Device Type .....	869
Generation Shift Factor Sensitivities.....	871
<b>Chapter 12: Time Step Simulation .....</b>	<b>872</b>
Time Step Simulation .....	873
Time Step Simulation: Quick Start .....	874
Time Step Simulation Dialog .....	876
Time Step Simulation Toolbar .....	878
Time Step Simulation Pages.....	879
Hourly Summary Page.....	880
Hourly Summary Page: Local Menu .....	882
Input Page .....	883
Matrix Grids.....	884
Results: Constraints Page .....	885
Binding Elements Dialog .....	888
Results Page .....	889
Custom Results Selection Dialog.....	890
Results Grid Pages .....	891
Time Step Simulation Options .....	892
Specifying and Maintaining a List of Timepoints.....	895
New Timepoint Dialog .....	896
Change Timepoint Time Dialog .....	897
TSB Case Description Page .....	898
Insert/Scale Column Dialog.....	899
Loading Hourly Input Data.....	901
Setting up Scheduled Input Data.....	902
Schedule Dialog.....	903
Schedules Page .....	905
Schedule Subscription Dialog .....	906
Schedule Subscriptions Page.....	908
Running a Timed Simulation .....	909
Running OPF and SCOPF Time Step Simulations .....	910
Storing Input Data and Results .....	913

<b>Chapter 13: PowerWorld Simulator Add-On Tools</b>	<b>914</b>
Introduction to Simulator Add-On Tools	915
PowerWorld Simulator PV/QV Overview	916
PV/QV PV Curves	917
PV/QV PV Curve Dialog	918
PV/QV Setup	919
PV/QV Options to Control the Transfer Increase	920
PV/QV Quantities to Track	923
PV/QV Limit Violations	925
PV/QV PV Output	926
PV/QV PV Results	928
PV/QV QV Setup	929
PV/QV QV Curves	931
PV/QV QV Curve Buses	932
PV/QV QV Curve Options	933
PV/QV QV Curve Results	934
PV/QV Refine Model	935
PowerWorld Simulator Optimal Power Flow Overview	936
OPF Objective Function	937
OPF Equality and Inequality Constraints	938
OPF Equality Constraints	939
OPF Inequality Constraints	940
Determining Set of Active Inequality Constraints	942
OPF Unenforceable Constraints	943
OPF Marginal Costs	944
OPF Primal LP	945
OPF Future Enhancements	946
OPF Options and Results	947
OPF Options: Common Options	948
OPF Options: Constraint Options	950
OPF Options: Control Options	952
OPF Options: Advanced Options	955
OPF Options: Solution Summary	956
OPF Options: All LP Variables	959
OPF Options: LP Basic Variables	960
OPF Options: LP Basis Matrix	961
OPF Options: Bus MW Marginal Price Details	962
OPF Options: Bus MVAR Marginal Price Details	963
OPF Options: Bus Marginal Controls	964
OPF Options: Inverse of LP Basis	965

OPF Area Records .....	966
OPF Bus Records .....	968
OPF Generator Records .....	969
OPF Interface Records .....	972
OPF Nomogram Records.....	973
OPF Line/Transformer Records.....	974
OPF DC Lines Records .....	975
OPF Load Records.....	976
OPF Super Area Records.....	978
OPF Controls .....	980
OPF Phase Shifter Records.....	981
OPF Example - Introduction .....	982
OPF Example - Marginal Costs.....	985
OPF Example - Super Areas .....	986
OPF Example - Enforcing Line MVA Constraints.....	988
Security Constrained Optimal Power Flow Overview .....	990
SCOPF Objective Function .....	991
SCOPF Dialog .....	992
SCOPF Solution Process.....	993
SCOPF Results.....	994
SCOPF Equality and Inequality Constraints .....	995
SCOPF Equality Constraints.....	996
SCOPF Inequality Constraints.....	997
SCOPF Control.....	999
SCOPF CTG Violations .....	1002
SCOPF LP Solution Details .....	1004
SCOPF All LP Variables .....	1005
SCOPF LP Basic Variables .....	1007
SCOPF LP Basis Matrix.....	1008
SCOPF Bus Marginal Price Details .....	1009
SCOPF Bus Marginal Controls.....	1010
SCOPF Example: Introduction .....	1011
SCOPF Example: Marginal Prices .....	1014
SCOPF Example: Unenforceable Constraints.....	1015
Available Transfer Capability (ATC) Analysis .....	1016
Available Transfer Capability Dialog .....	1017
ATC Dialog: Options Tab .....	1019
Transfer Result Reporting Options.....	1026
ATC Dialog: Analysis Tab.....	1028
ATC Dialog: Result Tab .....	1030

ATC Extra Monitors Dialog .....	1031
Multiple Scenario Available Transfer Capability Dialog .....	1033
Multiple Scenario ATC Dialog: Scenarios Tab .....	1034
Multiple Scenario ATC Dialog: Results Tab .....	1036
Multiple Scenario ATC Analysis - Results Tab: Local Menu Options .....	1038
Transfer Limiters Display .....	1039
ATC Analysis Methods - Solution Methods .....	1042
ATC Analysis Methods - Single Linear Step (SL) .....	1043
ATC Analysis Methods - Iterated Linear Step (IL).....	1044
ATC Analysis Methods - Iterated Linear Step (IL) then Full CTG Solution.....	1046
Simulator Automation Server (SimAuto) Overview .....	1049
Installing Simulator Automation Server .....	1050
Including Simulator Automation Server Functions .....	1051
Connecting to Simulator Automation Server .....	1052
Passing Data to the Simulator Automation Server .....	1054
Getting Data from the Simulator Automation Server.....	1055
Simulator Automation Server Functions.....	1056
ChangeParameters Function .....	1057
ChangeParametersSingleElement Function .....	1058
ChangeParametersSingleElement Sample Code .....	1059
ChangeParametersMultipleElement Function .....	1061
ChangeParametersMultipleElement Sample Code.....	1062
ChangeParametersMultipleElementFlatInput Function .....	1064
ChangeParametersMultipleElementFlatInput Sample Code .....	1065
CloseCase Function .....	1067
CloseCase Function: Sample Code .....	1068
GetFieldList Function .....	1069
GetFieldList Function: Sample Code .....	1070
GetParametersSingleElement Function.....	1071
GetParametersSingleElement Function: Sample Code Borland® Delphi.....	1073
GetParametersSingleElement Function: Sample Code Matlab® .....	1075
GetParametersSingleElement Function: Sample Code Microsoft® Visual Basic for Excel .....	1076
GetParametersMultipleElement Function .....	1078
GetParametersMultipleElement Sample Code Borland® Delphi.....	1080
GetParametersMultipleElement Function: Sample Code Matlab®.....	1081
GetParametersMultipleElement Function: Sample Code Microsoft® Visual Basic for Excel .....	1082
GetParametersMultipleElementFlatOutput Function .....	1084
GetParameters Function .....	1085
ListOfDevices Function .....	1086



ListOfDevices Function: Sample Code for Borland® Delphi .....	1088
ListOfDevices Function: Sample Code for Matlab®.....	1089
ListOfDevices Function: Sample Code for Microsoft® Visual Basic for Excel .....	1090
ListOfDevicesAsVariantStrings Function.....	1092
ListOfDevicesFlatOutput Function .....	1093
LoadState Function .....	1094
LoadState Function: Sample Code .....	1095
OpenCase Function .....	1096
OpenCase Function: Sample Code .....	1097
ProcessAuxFile Function .....	1098
ProcessAuxFile Function: Sample Code .....	1099
RunScriptCommand Function .....	1100
RunScriptCommand Function: Sample Code .....	1101
SaveCase Function.....	1102
SaveCase Function: Sample Code.....	1103
SaveState Function .....	1104
SaveState Function: Sample Code .....	1105
SendToExcel Function .....	1106
SendToExcel Function: Sample Code .....	1107
WriteAuxFile Function.....	1108
WriteAuxFile Function: Sample Code.....	1109
Simulator Automation Server Properties .....	1111
ExcelApp Property.....	1112
ExcelApp Property: Sample Code.....	1113
CurrentDir Property.....	1114
CurrentDir Property: Sample Code.....	1115
ProcessID Property .....	1116
ProcessID Property: Sample Code .....	1117
RequestBuildDate Property .....	1118
PowerWorld Object Variables .....	1119
Simulator Automation Server (version 9).....	1121
Installing Simulator Automation Server (version 9) .....	1122
Including Simulator Automation Server Functions (version 9).....	1123
Connecting to Simulator Automation Server (version 9).....	1124
Simulator Automation Server Properties (version 9) .....	1126
ExcelApp Property (version 9) .....	1127
Simulator Automation Server Functions (version 9).....	1128
ChangeParameters Function (version 9).....	1129
CloseCase Function (version 9) .....	1130
GetParameters Function (version 9).....	1131

ListOfDevices Function (version 9).....	1132
LoadContingencies Function (version 9) .....	1133
OpenCase Function (version 9) .....	1134
ProcessAuxFile Function (version 9).....	1135
RunScriptCommand Function (version 9) .....	1136
SaveCase Function (version 9) .....	1137
SendToExcel Function (version 9).....	1138
WriteAuxFile Function (version 9).....	1139
PowerWorld Object Variables (Version 9) .....	1140
<b>Chapter 14 : Auxiliary Script/Data Files.....</b>	<b>1141</b>
Auxiliary Files .....	1142
Script Command Execution Dialog .....	1144
Quick Auxiliary Files Dialog .....	1145
Script Section .....	1146
Script General Actions .....	1148
Script Edit Mode Actions.....	1161
Script Run Mode Actions.....	1167
Script PowerFlow Submode Actions .....	1174
Script Contingency Submode Actions .....	1178
Script ATC Submode Actions.....	1180
Script Fault Submode Actions.....	1182
Script PV Submode Actions.....	1183
Script QV Submode Actions .....	1186
Data Section.....	1188
Data Argument List.....	1189
Data Key Fields.....	1192
Data List .....	1193
Data SubData Sections.....	1194
Data ATC_Options .....	1195
Data ATCExtraMonitor .....	1196
Data ATCScenario .....	1197
Data AuxFileExportFormatData .....	1199
Data AuxFileExportFormatDisplay .....	1200
Data BGCalculatedField .....	1201
Data Bus.....	1202
Data BusViewFormOptions.....	1203
Data ColorMap.....	1204
Data Contingency .....	1205
Data CTG Options .....	1211
Data CTGElementBlock.....	1212

Data CustomCaseInfo.....	1213
Data CustomColors .....	1214
Data DataGrid .....	1215
Data DynamicFormatting .....	1216
Data Filter .....	1218
Data Gen .....	1220
Data GeoDataViewStyle.....	1221
Data GlobalContingencyActions .....	1223
Data HintDefValues.....	1224
Data InjectionGroup.....	1225
Data Interface .....	1226
Data LimitSet .....	1228
Data Load .....	1229
Data LPVariable .....	1230
Data ModelCondition .....	1231
Data ModelExpression .....	1232
Data ModelFilter .....	1233
Data MTDCRecord .....	1234
Data MultiSectionLine.....	1238
Data Nomogram .....	1239
Data Owner.....	1240
Data PieChartGaugeStyle.....	1242
Data PostPowerFlowActions .....	1243
Data PWCaseInformation.....	1244
Data PWFormOptions.....	1245
Data PWLPOPFGTGViol.....	1246
Data PWLPTabRow .....	1247
Data PWPVResultListContainer .....	1248
Data QVCurve .....	1249
Data QVCurve_Options.....	1250
Data SelectByCriteriaSet.....	1251
Data ShapeFileExportDescription .....	1253
Data StudyMWTransactions .....	1254
Data SuperArea .....	1255
Data TSSchedule .....	1256
Data View .....	1257
Data Section for Display Auxiliary File .....	1258
Data Argument List for Display Auxiliary File .....	1259
Data Key Fields for Display Auxiliary File .....	1261
Data List for Display Auxiliary File.....	1262

Special Data Sections .....	1263
Data Subdata Sections for Display Auxiliary File.....	1264
Data Common Sections .....	1265
Data DisplayLine.....	1266
Data PieChartGaugeStyle.....	1267
Data View .....	1268
<b>Chapter 15 : PowerWorld Simulator Project Files.....</b>	<b>1269</b>
Overview of PowerWorld Simulator Project Files .....	1270
PowerWorld Project Initialization Script.....	1271
Associating Project Files with Simulator.....	1272
Creating a New Project File.....	1273
Create Project Dialog .....	1274
Opening an Existing Project .....	1276
<b>Chapter 16 : Tutorials.....</b>	<b>1277</b>
Tutorial: Creating a New Case Page 1 of 13 .....	1278
Tutorial: Inserting a Bus Page 2 of 13.....	1279
Tutorial: Inserting a Generator Page 3 of 13 .....	1281
Tutorial: Saving the Case Page 4 of 13 .....	1283
Tutorial: Entering a Second Bus with Load Page 5 of 13.....	1284
Tutorial: Inserting a Transmission Line Page 6 of 13 .....	1287
Tutorial: Inserting a Line Flow Pie Chart on a Transmission Line Page 7 of 13 ..	1289
Tutorial: Inserting Circuit Breakers Page 8 of 13.....	1291
Tutorial: Inserting a Transformer Page 9 of 13 .....	1293
Tutorial: Inserting a Switched Shunt Page 10 of 13.....	1296
Tutorial: Inserting Text, Bus and Line Fields Page 11 of 13.....	1298
Tutorial: Solving the Case Page 12 of 13.....	1302
Tutorial: Adding a New Area Page 13 of 13 .....	1304
Starting with an Existing Case Page 1 of 15.....	1307
Tutorial: Loading an Existing Power Flow File Page 2 of 15.....	1308
Tutorial: Case Information Displays Page 3 of 15.....	1309
Tutorial: Solving the Case Page 4 of 15 .....	1310
Tutorial: Building Onelines Page 5 of 15.....	1311
Tutorial: Entering a Bus Page 6 of 15 .....	1312
Tutorial: Automatic Line Insertion Page 7 of 15 .....	1313
Panning and Zooming Page 8 of 15 .....	1314
Tutorial: Adding Background Page 9 of 15 .....	1315
Tutorial: Simulating the Case Page 10 of 15.....	1316
Tutorial: Run-time Object Dialogs Page 11 of 15.....	1317
Online Local Menu Page 12 of 15.....	1318
Tutorial: Area Page 13 of 15.....	1319

Limit Violations Page 14 of 15 .....	1320
Other Case Information Displays Page 15 of 15 .....	1321
Tutorial: Solving an OPF Page 1 of 6 .....	1322
Tutorial: OPF Three Bus Example Page 2 of 6 .....	1323
Tutorial: OPF Line Limit Enforcement Page 3 of 6 .....	1324
Tutorial: OPF LMP Explanation Page 4 of 6.....	1325
Tutorial: OPF Marginal Cost of Enforcing Constraints Page 5 of 6 .....	1326
Tutorial: OPF Unenforceable Constraints Page 6 of 6 .....	1327
Tutorial: Contingency Analysis .....	1329



---

## **Chapter 1: Getting Started With PowerWorld**

---

This chapter provides the essential information you need to start using PowerWorld Simulator.

The following material is included:

- About this Manual
- Introduction to PowerWorld Simulator
- Introduction to Simulator Add-on Tools
- What's New in Version 13
- Getting Help
- PowerWorld Interface
- Toolbars

---

**About this Manual**

---

PowerWorld Simulator includes comprehensive, context-sensitive on-line help in addition to a printed manual. Using the on-line help is strongly recommended, since it provides features not available in a printed manual, such as the ability to jump to another topic. Furthermore, the on-line help is likely to be the more up-to-date reference as program updates are issued.



---

## Introduction to PowerWorld Simulator

---

PowerWorld® Simulator (Simulator) is a power system simulation package designed from the ground up to be user-friendly and highly interactive. Simulator has the power for serious engineering analysis, but it is also so interactive and graphical that it can be used to explain power system operations to non-technical audiences. With [Version 13](#) we've made Simulator more powerful, more visual, and easier to use.

Simulator consists of a number of integrated products. At its core is a comprehensive, robust Power Flow Solution engine capable of efficiently solving systems of up to 100,000 buses. This makes Simulator quite useful as a stand-alone power flow analysis package. Unlike other commercially available power flow packages, however, Simulator allows the user to visualize the system through the use of full-color [animated oneline diagrams](#) complete with [zooming and panning](#) capability. System models can be either modified on the fly or built from scratch using Simulator's full-featured graphical case editor. Transmission lines can be switched in (or out) of service, new transmission or generation can be added, and new transactions can be established, all with a few mouse clicks. Simulator's extensive use of graphics and animation greatly increases the user's understanding of system characteristics, problems, and constraints, as well as of how to remedy them.

The base package of Simulator is capable of solving power systems comprised of up to 100,000 buses. The base package also contains all the tools necessary to perform integrated economic dispatch, area transaction economic analysis, power transfer distribution factor (PTDF) computation, short circuit analysis, and contingency analysis. All of the above features and tools are easily accessible through a consistent and colorful visual interface. These features are so well integrated that you will be up and running within minutes of installation.

In addition to the features of the base Simulator package, various add-on tools are available. Please see [Introduction to Simulator Add-On Tools](#) for more information.

---

## Introduction to Simulator Add-On Tools

---

In addition to the features of the base Simulator package, various add-on tools are available. A brief introduction to each follows:

### **Voltage Adequacy and Stability Tool (PVQV)**

The purpose of the PVQV add-on is to allow the user to analyze the voltage stability characteristics of a system. After the PVQV simulation is complete, the user can graph various system parameters. For more information, see the [PVQV Overview](#).

### **Optimal Power Flow Tool (OPF)**

The purpose of an OPF is to minimize an [objective \(or cost\) function](#). In Simulator OPF the Linear Programming OPF algorithm (LP OPF) determines the optimal solution by iterating between solving a standard power flow and solving a linear program to change the system controls thereby removing any limit violations. For more information see the [OPF Overview](#).

### **Security Constrained Optimal Power Flow Tool (SCOPF)**

The OPF tool minimizes an objective function (usually total operation cost) by changing different system controls while meeting power balance constraints and enforcing base case operating limits. The SCOPF tool takes it one step further by considering contingencies that may arise during system operation and ensuring that in addition to minimizing the objective function, no unmanageable contingency violations occur. For more information see the [SCOPF Overview](#).

### **Available Transfer Capability Analysis Tool (ATC)**

ATC analysis determines the maximum MW transfer possible between two parts of a power system without violating any limits. For more information see the [ATC Analysis Overview](#).

### **Transient Stability (TS)**

Transient Stability analysis allows the analysis of system dynamic response to a fault on the system. *The Transient Stability module has not yet been released with version 13.*

### **PowerWorld Simulator Automation Server (SimAuto)**

SimAuto provides PowerWorld customers the ability to access PowerWorld Simulator functionality within a program written externally by the user. The Simulator Automation Server acts as a COM object, which can be accessed from various programming languages that have COM compatibility. Examples of programming tools with COM compatibility are Borland® Delphi, Microsoft® Visual C++, Microsoft® Visual Basic, and Matlab® (among others). For more information on SimAuto, see the [SimAuto Overview](#).

---

## What's New

---

Web links to previous version information: [Version 12](#) [Version 11](#) [Version 10](#) [Version 9](#) [Version 8](#) [Version 7](#) [Version 6](#)

Simulator Version 13 contains a number of major new features and several smaller enhancements designed to improve the performance and convenience of the package.

- User Interface
  - o New [Model Explorer](#) allows fast navigation through all case information displays without needing to open multiple windows.
  - o Toolbars and Menus have been replaced with Microsoft's new [Ribbon](#) interface to group related actions together for fast navigation and selection.
  - o Generally all the user interface dialogs now obey the Windows XP and Windows Vista Themes (look and feel) settings.
- Auxiliary Files and Display Auxiliary Files
  - o New [Export Format Description](#) allows quick creation of auxiliary files with specified object types and fields. Each object type may be custom filtered.
  - o Modified reading of an AUX file to now allow the creation of a new object if a required field is specified, but the required field is blank. Also, added better error messaging to the log when an object has blank values for required fields to better alert the user of the problem.
  - o Added support for several kinds of objects that previously could only be defined using SUBDATA sections.
    - ContingencyElement objects represent the <SubData CTGElement> sections for all Contingency objects.
    - ViolationCTG objects represent the <SubData LimitViol> sections for all Contingency objects.
    - CTGElementBlockElement objects represent the <SubData CTGElement> sections for all CTGElementBlock objects.
    - GlobalContingencyActionsElement objects represent the <SubData CTGElement> section for the GlobalContingencyActions object.
    - PostPowerFlowActionsElement objects represent the <SubData CTGElement> section for the PostPowerFlowActions object
    - InterfaceElement objects represent the <SubData InterfaceElement> sections for all Interface objects.
    - PartPoint objects represent the <SubData PartPoint> sections for all Injection Group objects.
    - ReactiveCapability objects represent the <SubData ReactiveCapability> sections for all Gen objects.
    - MTDCBus, MTDCConverter, MTDCTransmissionLine objects represent their respective SubData sections for all MTDCRecord objects.
  - o Added support for calling specific sub-types of PartPoints via script commands. These include PartPointGen, PartPointInjectionGroup, PartPointLoad, and PartPointShunt. These special object types then give you access to a combination of all the PartPoint fields and specific device fields simultaneously.
  - o Added support for calling specific sub-types of InterfaceElements via script commands. These include InterfaceElementBranch, InterfaceElementBranchClose, InterfaceElementBranchOpen, InterfaceElementDCLine, InterfaceElementGen, InterfaceElementInjectionGroup, InterfaceElementInterface, InterfaceElementLoad, and InterfaceElementMSLine. These special object types then give you access to a combination of all the InterfaceElement fields and specific device fields simultaneously.

- o Added the ability to store the Advanced Sort as part of the DataGrid Auxiliary file section.
- Auxiliary File SCRIPT and SimAuto Changes
  - o Changes throughout to support the use of object [labels](#). If you give all your objects a unique label then the same auxiliary files can work for multiple cases regardless of bus numbering.
  - o New and modified script commands
    - DeleteIncludingContents(\*) Use this to delete objects of a particular type and other objects that these contain.
    - New optional parameter to the SaveData(\*) script command to specify sort order.
    - CalculateTLRMultipleElement(\*)
    - SaveDataWithExtra(\*)
    - SaveDataUsingExportFormat(\*)
    - ExitProgram
    - ExportOnlineAsShapeFile(\*)
    - CalculateRXBGFromLengthConfigCondType(\*)
    - DetermineShortestPath() and DeterminePathDistance()
    - New optional parameter "ApplyTransfer" to the ATCDetermineATCFor(\*) to set system state to the calculated transfer level.
- Available Transfer Capability (ATC) Enhancements
  - o Added better handling of degenerate cases in ATC such as when an injection group runs out of injection.
  - o Added a new option to iterate on a failed contingency. These options perform iterations of the power flow solution to find the transfer level at which the contingency fails.
- Case Information Displays
  - o Most Case Information Displays are viewed through the [Model Explorer](#).
  - o The right-click local menu of case information displays has been standardized to be consistent across all case information displays. The menu mimics the [Case Information Toolbar](#) directly.
  - o New option to "Send to Open Office" which mimics the "Send to Excel" feature.
  - o There is now a new "Is Blank" and "Not is Blank" condition comparison for [Advanced Filters](#).
  - o Added option to highlight rows if their Selected? field is YES.
  - o Added option to show a hint describing the field when you hover over a column heading.
  - o Display/Column Options
    - Available fields are grouped in folders by category making the navigation of the large number of available fields much easier.
    - Special folders are available providing you an easy summary of the key and required fields
    - Option to show variable names alongside traditional field names
    - Fields identified by data type (string, integer, floating point)
    - Automatically show hints to better let you know what the fields actually show.
- Connection Analysis Tools (Graph Theory)
  - o Added two new tools [Determine Path Distances to Buses...](#) and Determine Shortest Path Between which help calculate topological distances between parts of the network. These tools allow you to define a specific distance measure for each branch (X, Z, Length, number of Nodes, or a specified field), and also operating on Buses, Substations, Areas, Zone, Super Areas and Injection Groups.
- Contingency Analysis Enhancements

- o Added ability to view all contingency definitions in a single combined table case information display giving you possibility of using a single DATA section to define contingencies, or even using copy/paste to/from a spreadsheet to define your contingencies.
- o Added ability to view all contingency violations in a single combined table case information display. Also gives ability to define using a DATA section or using copy/paste to/from a spreadsheet.
- o Pan to object on oneline when selecting a limit violation in the contingency tool.
- Data Objects
  - o There is now no limit on the number of [Custom Expressions](#) defined for each object type.
  - o Added ability to specify the number of custom floating points, integers, and strings that each type of object has using the [Custom Field Descriptions](#). You can also give names and column headings to the fields.
  - o New supplemental data object type.
  - o Improved storage of interfaces to accommodate large numbers of interfaces.
- Dialogs
  - o Most device dialogs now have a Custom Tab which contains access to all custom singles, floating points, and integer values, as well as the Memo and Selected? field for each object.
  - o Added the Branches Tab to the Multi-Section Line Information Dialog to list all the individual branches in the multi-section line.
  - o On the Equivalencing Dialog, a feature was added to "Merge Shunts" across low impedance branches. This was done because we have continued to see strange input data with very large bus shunt values at either end of low impedance branches. (For example +10,000 MW at the FROM end and -10,000 MW at the TO end). The Merge Shunts button helps clean up this data.
- File Formats
  - o PowerWorld Binary File (PWB) version 13
    - Option to perform the case autosave in Run Mode.
  - o PowerWorld Auxiliary File (AUX) support
    - User prompt when saving the [Limit Monitoring Settings](#) from the Limit Monitoring Settings dialog to determine whether to use primary or secondary key fields or labels for identifying elements.
    - Extensive support added for writing to auxiliary file using object [labels](#). If you define labels for all your important devices such as generator, loads, and branches and these labels do not change between models, then you can create AUX files that work with different topologies.
  - o PTI RAW file support
    - Allow opening a RAW file with options instead of opening with defaults only.
  - o GE EPC file support
    - Automatically set the option for sharing of generator vars across groups of buses during remote regulation appropriately when reading an EPC file.
- [Geographic Data Views](#)
  - o Quickly create graphical representations of buses, generators, loads, switched shunts, substations, areas, zones, super areas, transmission lines, and injection groups.
  - o Latitude and longitude coordinates specified with buses and substations are used to place the objects geographically on the display. (The latitude and longitude information must be specified in order to use this feature.)
  - o Object attributes such as color, size, rotation rate, rotation angle, and visibility can easily be formatted based on element field values.

- o Different element types can be combined on a common display by using a previously saved PWD file as a starting template.
  - o Geographic borders can be included on the display when inserting new objects.
- Online Diagrams
  - o Completed the support for viewing of all objects on the online diagram in 3D mode. Also enhanced the ability to specify 3D Stacks and Stack Slices.
  - o Enhanced GIS Support
    - Identify closest facilities to a selected point.
    - [Geographic data view](#).
  - o Can now [select by region](#) using an ellipse or a polygon. Also modified so that holding the CTRL key forces the Ellipse or Rectangle region selection to be centered on the initial click. Also modified so that holding the SHIFT key forces the Ellipse or Rectangle region selection to be a Circle or Square.
  - o Added new Copy Format and Paste Format feature.
  - o Allow field prefixes to be edited for selected objects.
  - o Contouring: Dialog only shows object types that occur on the online.
  - o Rotate through onelines specified in the keyboard shortcuts in a "slideshow".
  - o Added more shapes for the display of breakers.
  - o Added ability to specify a generator field which appears inside each generator symbol.
  - o Added new GIS Tool for clicking on a oneline and returning a list of the closest facilities to that point
- Optimal Power Flow Enhancements
  - o New Add-on Tool to simulate Ancillary Services Reserve Markets.
  - o Generator and Load reserve controls include spinning, regulating, and supplemental reserves.
  - o Area and Zonal constraints include regulating, contingency and operation reserves modeled by demand curves.
- Power Flow Solution
  - o Added better ability to handle areas that span multiple islands when using area interchange.
  - o Allow specification of metered end for two-terminal and multi-terminal DC line records.
  - o Under rare circumstances it's possible for a transformer controlling it's FROM bus to have a negative voltage to tap sensitivity. It doesn't make sense for the transformer to attempt to change its tap in this situation, therefore Simulator will not anymore.
- PV and QV Curve Enhancements
  - o Added new options for exporting results to a text file.
  - o For critical scenarios that are critical because an inadequate voltage was found, we now report the bus and voltage of the lowest inadequate voltage.
- Sensitivity Calculations
  - o When using a Super Area as a buyer or seller in the ATC, PTDF, TLR, and voltage sensitivities the area participation factors defined for the super areas are used to calculate appropriate sensitivities.
- Time Step Simulation Enhancements
  - o Added support for auxiliary file commands to run, stop and solve single points.
  - o Added state reset when resetting the simulation.
  - o Added support for schedule subscriptions for interfaces.
  - o Time format throughout Simulator is now compatible with the Windows locale (regional settings).

---

## Help

---

On-line help is available in PowerWorld Simulator via the **Help** ribbon group on the [Window](#) ribbon tab, or by pressing the **F1** key on many dialogs and displays. Context-sensitive help is available on the oneline diagrams. To obtain object specific help, position the mouse over the object in question on the oneline and press the **F1** key.

Sample power flow cases and other information are available at the PowerWorld web site:  
<http://www.powerworld.com/>

Contact technical support at [support@powerworld.com](mailto:support@powerworld.com) for answers to your questions regarding any PowerWorld product. Or call us at (217) 384-6330.

**PowerWorld Corporation**

2001 S. First Street  
Suite 203  
Champaign, IL 61820

---

**Contact Information**

---

PowerWorld Corporation  
2001 South First Street  
Champaign, IL 61820

Phone: (217) 384-6330  
Fax: (217) 384-6329  
Toll Free: (877) 748-7840 - U.S. and Canada only

General Information: [info@powerworld.com](mailto:info@powerworld.com)  
Technical Support: [support@powerworld.com](mailto:support@powerworld.com)  
Sales: [sales@powerworld.com](mailto:sales@powerworld.com)

[www.powerworld.com](http://www.powerworld.com)



---

## **Windows Basics**

---

Simulator 13 runs under all Windows 95 and later operating systems. Since much of the interaction between Simulator and the user is accomplished by using the mouse, we have designed the interface to obey consistent conventions for mouse usage. In general, the left-mouse button is used to affect some sort of immediate change or control over a power system element, while the right mouse button is used to gain more information about a power system element or to view a list of available options. More details on are provided throughout this manual.

---

## PowerWorld Simulator: Getting Started

---

The key to using Simulator is to recognize that it has two distinct modes **Edit Mode** and **Run Mode**. The Edit Mode is used to construct new simulation cases or to modify existing cases, while the Run Mode is used to perform the actual power system simulation. You can easily switch between the modes using the **Edit Mode** and **Run Mode** buttons in the **Mode** ribbon group.

If you are new to Simulator and seek a quick means of familiarizing yourself with it, we recommend starting with the tutorials; see [Creating a New Case](#) or [Starting with an Existing Case](#)

Sample cases are provided with the software in the "Sample Cases" directory. If you're interested in learning by doing, you may wish to just [open one of the sample cases](#) and try the different Simulator features.

---

## Edit Mode Introduction

---

The Edit Mode is used to create a new case or to modify existing cases. To switch to Edit Mode, click on the **Edit Mode** button in the **Mode** ribbon group on any of the ribbon tabs.

Here is a sampling of things you can do in Edit Mode:

- Create a new case; see [New Case](#) for details.
- Create a new oneline diagram; see [New Oneline](#) for details.
- Add new components graphically to an existing case; see [Draw](#) ribbon tab.
- Modify the appearance of the oneline objects; see [Formatting](#) ribbon group for details.
- View and modify a case using non-graphical lists displays; see [Case Information Displays](#) for details.
- Equivalence a case; see [Equivalencing](#) for details.
- Append a subsystem to an existing case; see [Appending a Case](#) for details.

For more details on the Edit Mode please see [Edit Mode Overview](#).

---

## Run Mode Introduction

---

The Run Mode is used to solve a single Power Flow Solution, run one of the available load flow tools, or run a time-domain simulation of the power system. To access the Run Mode, click on the **Run Mode** button in the **Mode** group of the Simulator Ribbon.

The key menu associated with the Run Mode is the [Simulation](#) menu. This menu allows you to perform a single Power Flow Solution (however, it is quicker to use the [Quick Access](#) toolbar).

Other key components of the Run Mode include:

- The online diagrams, which allow you to view the case graphically. See [Online Diagram Overview](#) for details.
- The Case Information Displays, which allow you to view the entire power system case using list displays. See [Case Information Displays](#) for details.
- Dialogs to change the simulation options and the Power Flow Solution. See [Simulation Options](#) for details.
- Scaling to allow easy variation in the load, shunts, and generation at any number of buses. See [Scaling](#) for details.
- Contouring, which shows a color contour representing the variation in any power system parameter across a system. See [Contouring](#) for details.
- Transfer distribution factor calculations. See [Power Transfer Distribution Factors](#) for details.
- Perform a fault analysis. See [Fault Analysis](#) for details.
- Run [Transfer Capability](#) studies.
- Perform an [Optimal Power Flow](#) (OPF) or [Security Constrained Optimal Power Flow](#) (SCOPF) analysis.
- Generate [PV and QV curves](#).

---

## Script Mode Introduction

---

The Script Mode is used to access the Script Command window. Simulator scripting allows a method of grouping multiple commands for sequential processing by Simulator. From the [Script Command Execution](#) window, the user can manually enter [script commands](#) for processing, or load an [auxiliary file](#) containing multiple script commands and data modification commands.

Some features in Simulator are available exclusively in either [RUN](#) mode or [EDIT](#) mode. This functionality is preserved in the script language, but with the addition of a **submode** feature. Submodes determine which script commands can be called. Only those commands available to the current submode can be executed. You will always be in one of the submodes when executing a script. If the Script Command Execution Dialog is opened from Edit Mode, Simulator defaults to the **EDIT**, **CASE** submode. If the Script Command Execution Dialog is opened from Run Mode (or when a script is initially started), Simulator defaults to the **RUN**, **POWERFLOW** submode.

To switch submodes, use the [EnterMode \(mode or submode\)](#) script command. The following list includes the submodes available to each of the Simulator modes. Click on the links below to see the actions available to each mode and/or submode.

### [Edit Mode](#)

- [Case Submode](#)

### [Run Mode](#)

- [PowerFlow Submode](#)
- [Contingency Submode](#)
- [ATC Submode](#)
- [Fault Submode](#)
- [PV Submode](#)
- [QV Submode](#)


There are also a number of general script actions that can be executed in any of the submodes. See [Script General Actions](#) for more details.

---

## Message Log

---

The Message Log displays detailed results of each Power Flow Solution, chronicling the solution process iteration by iteration. It also reports messages raised by Simulator in performing various operations, such as opening or validating a case. The Message Log can be helpful when you run into problems solving a particular simulation case. The Message Log is not used with Viewer.

To display the Message Log, go to the [Tools](#) ribbon tab, and click the button  **Log** on the **Log** ribbon group. This button is also available on the [Add Ons](#) ribbon tab.

Right-clicking on the Message Log displays its local menu. To change the font characteristics of the log, select **Change Font**. To suppress or change color of [certain messages associated with the power flow case](#) select either **Suppress Messages** or **Color Messages** and then the desired message whose characteristics want to change. You can find certain text by clicking on **Find**, and then entering the text in the dialog displayed. Also, you can highlight all the text in the log, or inversely unselect all the highlighted text by clicking on **Select All**, or **Unselect All**, respectively. To print or copy the contents to the Windows clipboard of a highlighted section of the message log, select **Print Selection** or **Copy Selection to Clipboard**. You can also clear the contents of the log by selecting **Clear Window**. Lastly, you will find some log options available, such as **Change Maximum Lines**, **Disable Logging**, **Disable Time Stamp**, and **Save Log to File** options which allow automatically saving the log to a file during load flow analysis.

---

## Custom and Memo Display

---

The Custom tab is now available on many dialogs and displays in Simulator. The purpose of the custom tab is to allow the user to add their own custom information and comments to data and objects in the load flow case. Simply switch to the Memo or Custom page on a display and start typing in the memo box, or add values or text in the custom fields. The information added to these memo pages is stored with the objects in the load flow case and saved to [PWB](#) files.

The custom fields section allows access to setting and changing the values for custom fields that have been defined for the bus. Defining custom fields is detailed in [Custom Field Descriptions](#).

The custom section also contains a **Selected?** check box. This is a special field that is associated with most objects in Simulator. In many of the places where filtering is allowed, filtering can be done based on the **Selected?** field. Additional information about this field can be found under the [Set Selected Field](#) topic.

The memo section of the dialog is simply a location to log information about the bus. Any information entered in the memo box will be stored with the case when the case is saved to a [PWB](#) file.

---

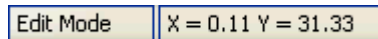
## Status Bar

---

The Status Bar is displayed across the bottom of the PowerWorld Simulator window. The left-most field of the status bar displays the current Simulator mode (Edit or Run). The remaining fields vary depending on which mode of operation Simulator is currently in. The status bar displays Tool Tips when the cursor is positioned over a toolbar button or menu item. The tool tip is also shown next to the cursor after a short time delay.

### Edit Mode

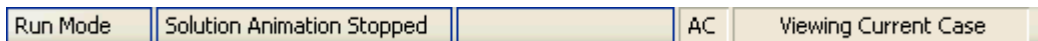
The Edit Mode status bar displays the [Screen Coordinates](#) of the cursor when positioned over a oneline diagram.



**Edit Mode Status Bar**

### Run Mode

The left side of the Run Mode status bar displays simulation status ("Solution Animation Stopped" or "Solution Animation Running"), AC or DC depending on solution options and the [Difference Flows](#) status ("Current Case", "Base Case", or "Difference Case").

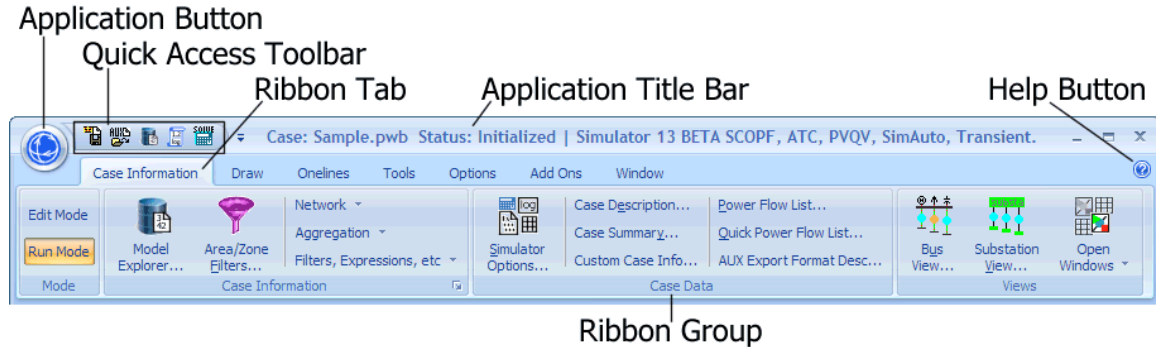


**Run Mode Status Bar (Left Side)**

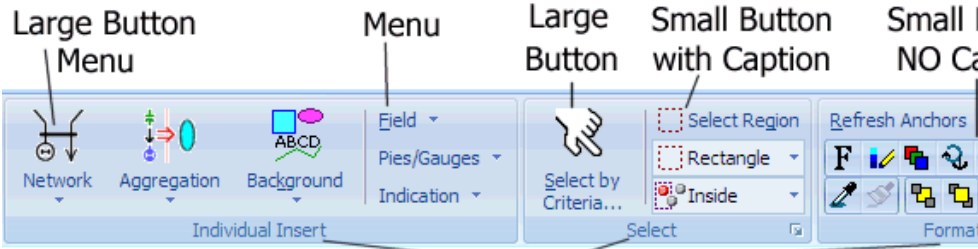





















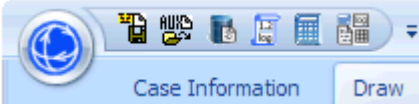

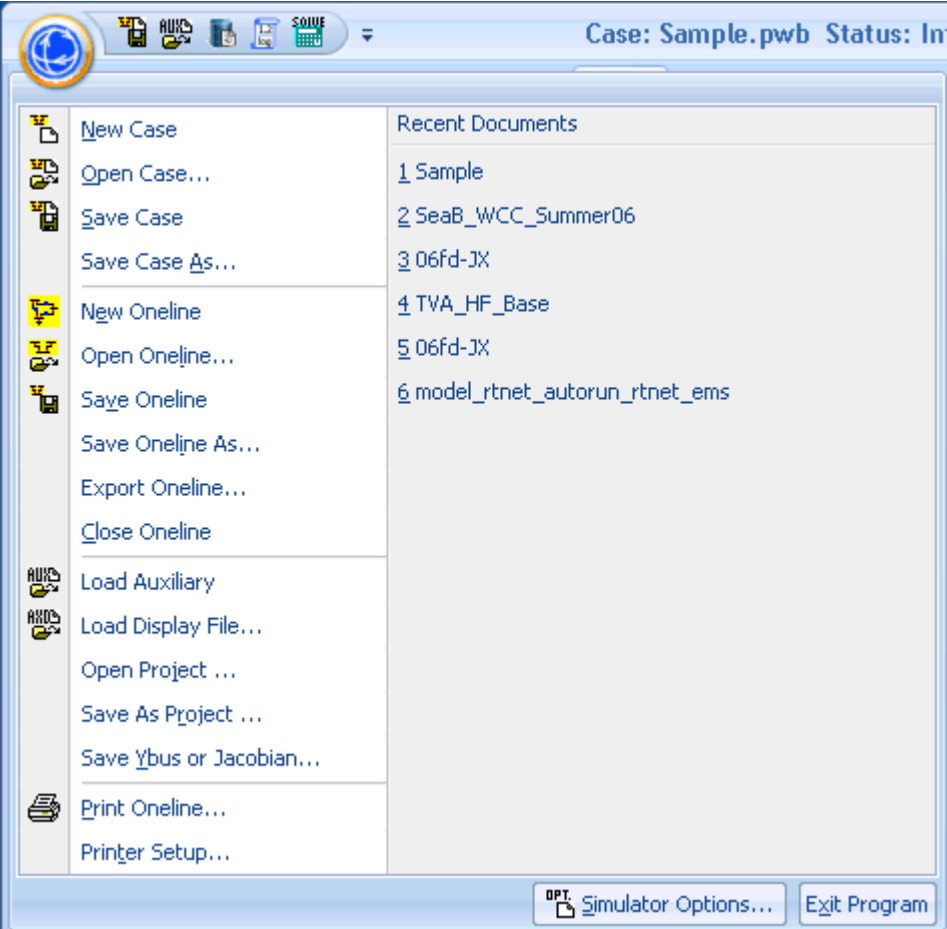
## Ribbons User Interface

PowerWorld's user interface consists of a ribbon across the top of the program. Important terminology regarding the ribbon is depicted in the images below.



Ribbon	The entire strip across the top is called the ribbon
Ribbon Tab	<p>The ribbon consists of several ribbon tabs which group together features of the software that are used together. There are seven ribbons which have their own purposes and own help topics listed as follows.</p> <ul style="list-style-type: none"> <li>• <a href="#">Case Information Ribbon Tab</a>: primarily used to navigate and look through all the data in your model.</li> <li>• <a href="#">Draw Ribbon Tab</a>: primarily used to draw new oneline diagrams or edit existing onelines by adding, moving, formatting, or resizing existing oneline objects. Most of the options on the <a href="#">Draw</a> ribbon tab are only available in <a href="#">Edit Mode</a>.</li> <li>• <a href="#">Onelines Ribbon Tab</a>: primarily used after you have already created a oneline diagram. This ribbon provides features for customizing the appearance of your oneline diagram.</li> <li>• <a href="#">Options Ribbon Tab</a>: all of the buttons on this ribbon are also available on one of the other ribbons, however this ribbon brings all the options in the software into one place.</li> <li>• <a href="#">Tools Ribbon Tab</a>: this ribbon provides access to all of the analysis tools that are available in the base package of PowerWorld Simulator. You will use this ribbon when you are performing power flow analysis, contingency analysis, or using the sensitivities tools for instance.</li> <li>• <a href="#">Add Ons Ribbon Tab</a>: this ribbon provides access to all the add-on tools available for Simulator including the OPF, SCOPF, ATC, and PVQV tools. If you have not purchased these tools then the options will be grayed out.</li> <li>• <a href="#">Windows Ribbon Tab</a>: this ribbon provides access to customizing the Windows in the User Interface. It also has some information reading help topics .</li> </ul> <p>The goal in designing these ribbons was to create several task specific ribbons which represent the various ways that the software is used. Our hope is that you will not spend a lot of time flipping back and forth between ribbons. If you find yourself doing this please contact PowerWorld Corporation to help us make the software better for the future.</p>

<b>Ribbon Group</b>	<p>Inside a particular ribbon tab, there are around 4 to 7 groupings called ribbon groups. The following image shows three ribbon groups (Individual Insert, Select, and Formatting) with various buttons and menus available.</p>  <p style="text-align: center;"><b>3 Ribbon Groups</b></p> <p>ribbon groups join sets of buttons and menus that are related to one another. The large buttons and icons are used to draw attention to the most commonly used features of the software. All features are still available through drop-down menus in the same manner as was available previously. Our design goal in creating and laying out the ribbons was to provide the new user with insight about using the software, while at the same time giving the experienced user quick access to common features.</p> <table border="1" data-bbox="410 850 821 1461"> <tbody> <tr> <td>Large Button</td><td></td></tr> <tr> <td>Large Button Menu</td><td></td></tr> <tr> <td>Small Button With Caption</td><td></td></tr> <tr> <td>Small Button Menu</td><td></td></tr> <tr> <td>Small Button without Caption</td><td></td></tr> <tr> <td>Menu Only Caption</td><td></td></tr> </tbody> </table>	Large Button		Large Button Menu		Small Button With Caption		Small Button Menu		Small Button without Caption		Menu Only Caption	
Large Button													
Large Button Menu													
Small Button With Caption													
Small Button Menu													
Small Button without Caption													
Menu Only Caption													
<b>Help Button</b> 	<p>Clicking the help button on the right edge of the ribbon brings up this online help</p>												
<b>Quick Access Toolbar</b>	<p>The quick access toolbar at the top left of the ribbon (near the Application Button) contains small buttons with no caption that are always visible. For more information about how to use and customize this see the <a href="#">Quick Access Toolbar</a> help.</p>												

	
Application Title Bar	Case: Sample.pwb Status: Initialized   Simulator 13 SCOPF, ATC, PVQV, SimAuto
Application Button	<p>Click on this to bring up the Application File Menu shown below. For more information about using the <a href="#">Application File Menu</a> see the <a href="#">Application File Menu</a> help.</p>  

---

## Quick Access Toolbar

---

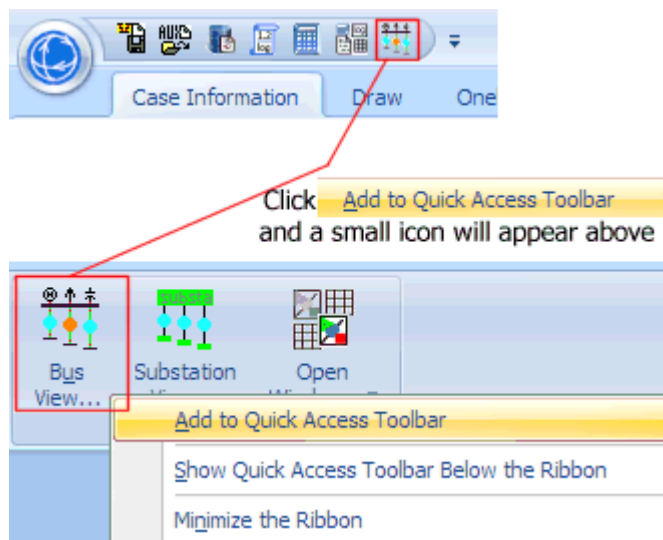
The Ribbon Quick Access Toolbar provides you with quick access to the most frequently used buttons. These buttons are found in the top left portion of the Ribbon just to the right of the [Application Button](#). The default buttons that appear in the Quick Access Toolbar are shown in the figure below.



The six default buttons are as follows from left to right.

- [Save Case](#)
- [Open Auxiliary File](#)
- [Model Explorer](#)
- [Message Log](#)
- [Single Solution](#)
- [Simulator Options](#)

The Quick Access Toolbar can also be customized very simply. If you find a button that you would like quicker access to, simply right-click on the button and then choose Add to Quick Access Toolbar. The small icon version of the button will then immediately appear up in the Quick Access Toolbar. This is depicted in the image below.

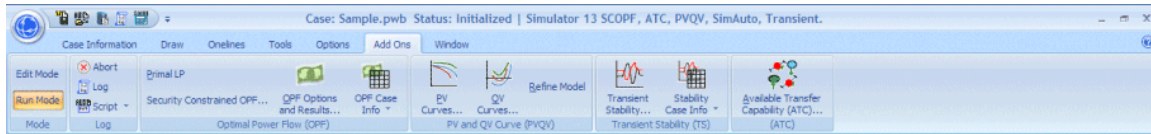


The Quick Access Toolbar can also be modified to show the Toolbar Below the Ribbon providing more space for specifying buttons.

Also experiment with Showing the Quick Access Toolbar below the Ribbon or Minimizing the Ribbon.

## Add Ons Ribbon Tab

The Add Ons ribbon consists of buttons and menus that access to the various Simulator Add On Tools. The ribbon is shown below when configured for the OPF, SCOPF, PVQV, ATC add on tools.



The Tools ribbon is broken into six ribbon groups

### Mode Ribbon Group

#### Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

#### Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

### Log Ribbon Group

#### Abort

Terminates the current Power Flow Solution. If the application is performing a timed simulation, pressing the abort button will pause the simulation. See [PowerWorld Simulation Control](#) for more details.

#### Log

Toggles the display of the [message log](#) window. The log window shows what is going on with the Power Flow Solution process and may prove useful when you are trying to track down a problem with a non-converging model.

#### Script

Opens the Script dialog, which can be used to call [script commands](#) or open [auxiliary files](#) containing script commands and data modifications. Note that the drop-down next to the Script button give convenient access to the [Quick Auxiliary Files](#).

### Optimal Power Flow (OPF) Ribbon Group

#### Primal LP

Click this button to solve the OPF. A more detailed description of what this does is found in the topic [OPF LP Primal](#).

#### Security Constrained OPF

Click this button to open the [Security Constrained OPF Dialog](#).

#### OPF Options and Results

Click this button to open the [OPF Options and Results Dialog](#).

#### OPF Case Info Menu

The OPF Case Info Menu provides direct access to several case information displays that show default columns related to input and output of the Optimal Power Flow. Instead of using this menu, PowerWorld encourages you leave the [Model Explorer](#)

open and navigate using the Explore Pane on the Model Explorer. All of these options are available under the Optimal Power Flow folder on the Model Explorer.



### **PV and QV Curve (PVQV) Ribbon Group**

#### **PV Curves**

Click this button to open the [PV Curves dialog](#).

#### **QV Curves**

Click this button to open the [QV Curves dialog](#).

#### **Refine Model**

Click this button to open the [Refine Model Dialog](#).

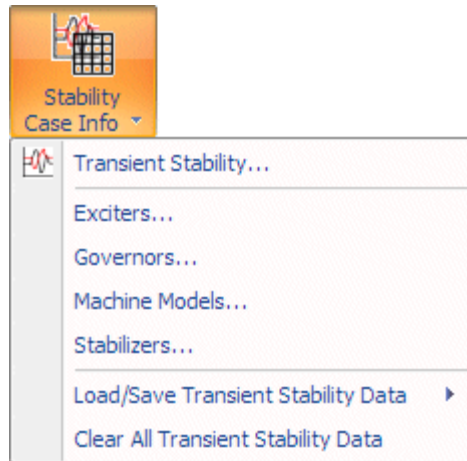
### **Transient Stability (TS) Ribbon Group**

#### **Transient Stability**

Click this button to open the Transient Stability Dialog.

#### **Stability Case Info Menu**

The OPF Case Info Menu provides direct access to several case information displays that show default columns related to input and output of the Transient Stability Tool.



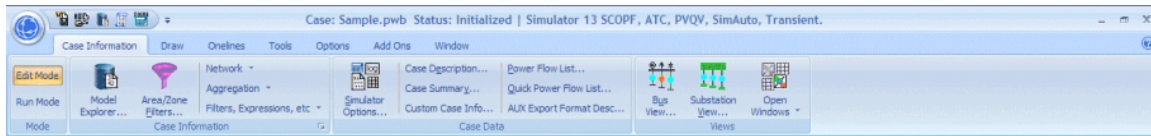
### **Available Transfer Capability Ribbon Group**

#### **Available Transfer Capability**

Click this button to open the [Available Transfer Capability Dialog](#).

## Case Information Ribbon Tab

The Case Information ribbon consists of buttons and menus that provide access to the information about the power system model. The ribbon is shown below



The Case Information ribbon is broken into four ribbon groups

### Mode Ribbon Group

#### Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

#### Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

### Case Information Ribbon Group

#### Model Explorer



Opens the [Model Explorer](#) giving access to all the [case information displays](#) for the model.

#### Area/Zone Filters

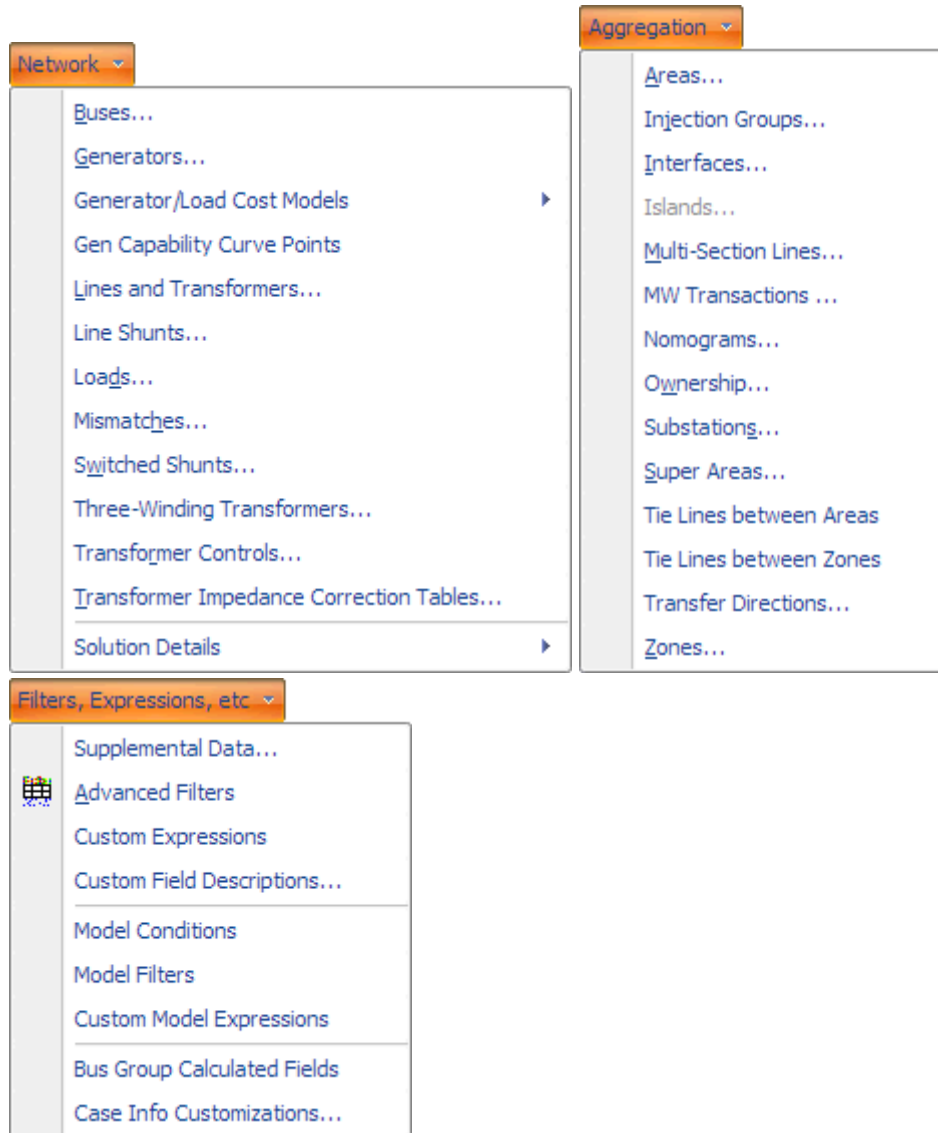


Opens the [Area/Zone/Owner Filters](#) display which allows filtering of the case information displays by Area, Zone, and Owner.

#### Menus for Network, Aggregation, and Filters, Expression, Etc

These three menus all perform the same action as opening the Model Explorer, except that they automatically set the Explore Pane of the Model Explorer to show the respective case information display. Instead of using these menus, PowerWorld encourages you leave the Model Explorer open and navigate using the Explore Pane on the Model Explorer.





## Case Data Ribbon Group

### Simulator Options

Opens the [Simulator Options Dialog](#).

### Case Description

Opens the [Case Description Dialog](#).

### Case Summary

Opens the [Case Summary Dialog](#).

### Custom Case Info

Opens the [Custom Case Information Display](#)

### Power Flow List

Opens the [Power Flow List](#)

### Quick Power Flow List

Opens the [Quick Power Flow List](#)

#### **AUX Export Format Desc**

Opens the [Auxiliary File Export Format Description Dialog](#) for Power System Objects.

### **Views Ribbon Group**

#### **Bus View**

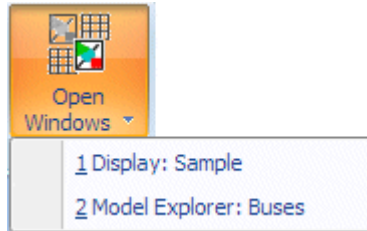
Opens the [Bus View Online](#) which provides a graphical display which allows you to quickly browse information about a bus and everything connect to that bus.

#### **Substation View**

Opens the [Substation View Online](#) which provides a graphical display which allows you to quickly browse information about a bus and everything connect to that substation.

#### **Open Windows Menu**

The Open Windows Menu provides a convenient method of switching between open windows. This includes User Interface Dialogs, Case Information Displays, and Online diagrams.

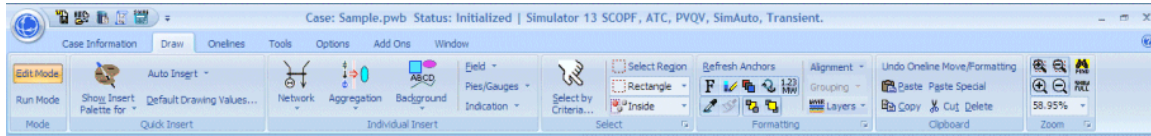


---

## Draw Ribbon Tab

---

The Draw ribbon tab is primarily used to draw new oneline diagrams or edit existing onelines by adding, moving, formatting, or resizing existing oneline objects. Most of the options on the Draw ribbon tab are only available in [Edit Mode](#). The Draw ribbon tab is shown below.



The Draw ribbon consists of seven ribbon groups described as follows

### Mode Ribbon Group

#### Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

#### Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

### Quick Insert Ribbon Group

The Quick Insert ribbon group contains buttons for creating a oneline diagram for a system model you have already read into Simulator. If you are building out your network (or even substation, area, or zone diagram), then using the Insert Palettes along with the Auto Insert routines is most efficient. For detailed help see the [Quick Insert](#) ribbon group. If you would instead like help on inserting a single object on your oneline diagram see the [Individual Insert](#) ribbon group topic.

### Individual Insert Ribbon Group

The Individual Insert ribbon group contains menus which provide access to buttons for inserting individual oneline objects to the oneline diagram. If you are building out your network (or even substation, area, or zone diagram), then it is much more efficient to use the [Quick Insert](#) ribbon group. However, for complete access to all objects including those that can not be "quick - inserted", see the help under the [Individual Insert](#) ribbon group topic.

### Select Ribbon Group

The Select ribbon group contains options to help select multiple oneline objects simultaneously. For more detailed help see the [Selection](#) ribbon group topic for more detailed help.

### Formatting Ribbon Group

The Format ribbon group contains options for formatting the selected objects on your active oneline. For more detailed help see the [Formatting](#) ribbon group topic for more detailed help.

### Clipboard Ribbon Group

The Clipboard ribbon group contains options for copying and pasting oneline display objects. For more detailed help see the [Clipboard](#) ribbon group topic for more detailed help.

### **Zoom Ribbon Group**

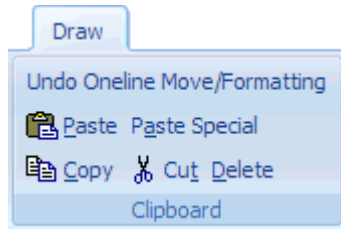
The Zoom ribbon group contains buttons for navigating a oneline diagram. For more detailed help see the [Zoom](#) ribbon group topic

---

## Clipboard Ribbon Group

---

The Clipboard ribbon group contains options for copying and pasting oneline objects while your are in [Edit Mode](#). It also provides access to the Undo drawing feature. This ribbon group is on the [Draw](#) ribbon tab. The Group is shown below.



### Undo

The Undo command is used in the edit mode to undo the last change made on the oneline diagram. The Undo command will only undo graphical changes, and will not undo any data changes in the power system model. See [Relationship Between Display Objects and the Power System Model](#) for information on the display/model relationships.

### Paste

The paste command copies the contents of the paste buffer (if any) onto the display at the current cursor location. Use the **Paste** command from the Edit Menu. Note that the paste buffer may contain both display objects and the underlying power system records. When pasting, the display objects are pasted regardless of whether an identical display object already exists on the oneline. In contrast, duplicate power system records are never pasted. This is because, for example, it is acceptable to have two display objects referring to the same generator, but the generator exists only as a single entity in the power system model. See [Relationship Between Display Objects and the Power System Model](#) for further details.

### Paste Special

Paste Special works the same as Paste, however it will also show a dialog asking the you to choose between

**Absolute Coordinates** : objects will be pasted using the exact same x/y coordinate as the copied objects had. This is useful when copying objects between two different oneline diagrams which contain the same geographic background.

**Coordinates Relative to Cursor** : objects will be pasted relative to cursor instead of using the exact x/y coordinates as the copied objects.

### Copy

The Copy Command copies the currently selected object(s) into the paste buffer without deleting them. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to copy just the display object(s), or copy both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To copy only the display object(s) and never the power system records from now on, select the **Always Copy Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

## Cut

The Cut Command is used in the Edit Mode to delete the currently selected object(s). To delete a set of objects, first select the objects. Then select the Cut command from the Edit Menu. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to delete just the display object(s), or delete both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To delete only the display object(s) and never the power system records from now on, select the **Always Delete Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Unlike Delete, Cut also copies the selection into the paste buffer.

## Delete

The Delete Command is used in the Edit Mode to delete the currently selected object(s). To delete a set of objects, first select the objects. Then select the Delete command from the **Edit** menu. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to delete just the display object(s), or delete both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To delete only the display object(s) and never the power system records from now on, select the **Always Delete Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Unlike [Cut](#), Delete does not copy the selection into the paste buffer.

---

## Formatting Ribbon Group

---

The **Formatting** ribbon group contains options for formatting the selected objects on your active oneline. This group is on the [Draw](#) ribbon tab and its options are available only in Edit mode. The options allow you to control such display object attributes as font, color, line styles, zoom-dependent visibility, and display layer level. The group is shown below.



A description of the actions on this group are as follows.

### Refresh Anchors

Invokes a command to [refresh anchors](#) on all oneline objects.



### Format Buttons

Clicking any of these five buttons will open the [Format Dialog](#). Each of the buttons opens the Format dialog to a specific tab on the Format Dialog, but they all ultimately open the same dialog.

#### Font

Select this option to change the font with which the selected objects are displayed. The [Font Tab](#) of the Format Selection Dialog is displayed.

#### Line/Fill

Select this option to change the line thickness and style with which a display object is drawn, or the fill color for closed shapes. The [Line/Fill Tab](#) of the Format Selection Dialog is displayed.

#### Levels/Layers

Select this option to change the stack level of an object, the layer the object is contained in, and optional settings for when an object should resize. The [Levels/Layers Tab](#) of the Format Selection Dialog is displayed.

#### Display/Size

Simulator can change the attributes of the way multiple objects are displayed using this option, which opens the [Display/Size page](#) of the Format Selection Dialog. The size of the objects can be adjusted, as well as the orientation of the object(s) from their terminal buses. Objects can also be Anchored to their terminal buses or devices, or marked as Immobile, meaning the object(s) cannot be moved on the oneline diagram.

#### Field

Select this option to change the attributes related to object text fields which are selected. The [Field Tab](#) of the Format Selection Dialog is displayed.



### Copy Format and Paste Format Buttons

The button on the left is the **Copy Format** button. After selecting an object this will copy all the format properties of that object into an internal buffer. When you then subsequently select other objects, you can click on the button on the right which is the **Paste Format** button. Clicking the **Paste Format** button will bring up the [Paste Format Dialog](#).



### Send to Back, Bring to Front Buttons

After selecting a group of oneline objects using any of the methods described on the [Select](#) ribbon group, you may then click the **Send to Back** button to cause all the objects selected to move to underneath other objects. Click the **Bring to Front** button to bring objects to the top.

The Send to Back and Bring to Front menu options govern the visibility of display objects that occupy the same screen stack level. All objects have an associated screen stack level, which may be one of Base, Background, Middle, or Top. Objects having a screen stack level of Top obscure all objects having stack levels of Middle, Background, or Base that occupy the same region of the screen. They may or may not obscure objects having the same stack level of Top depending on the order in which the objects having the same stack level are drawn. Selecting Send to Back for a selected object will cause it to be obscured by all other same-level objects that occupy its location. Selecting Bring to Front for a selected object will cause it to be drawn above all other same-level objects that occupy the same region of the screen. The Send to Back and Bring to Front menu options govern relative placement of objects only *within* stack levels. The Send to Back and Bring to Front options do not affect the relative placement of objects having different stack levels.

### Alignment Menu

The alignment menu provides access to the [Alignment Dialog](#) and direct access to options to align the selected objects as specified in the menu items.

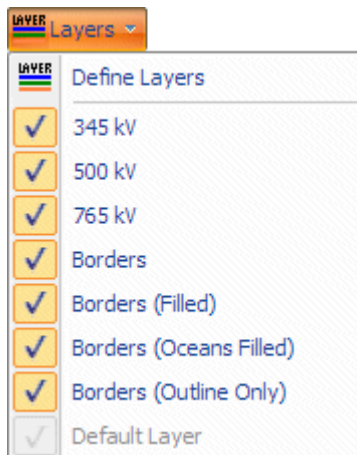
### Grouping Menu

Grouping is only available for background oneline objects (background lines, ellipses, text, etc...). This menu provides button to Group and Un-Group objects.

### Layers Menu

The Layers Menus provide access for opening the [Define Layers Dialog](#) for the presently active oneline diagram.

Underneath the button for defining layers is a list of the presently defined layers. You may check or uncheck the various layers to hide or show the objects in those layers.



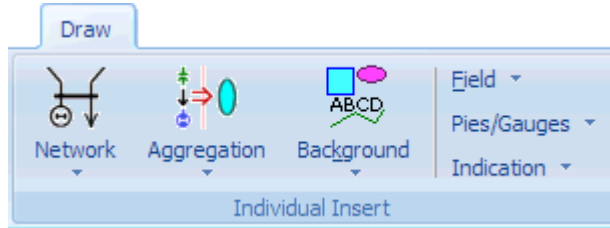


---

## Individual Insert Ribbon Group

---

The Individual Insert ribbon group contains menus which provide access to buttons for inserting individual oneline objects to the oneline diagram. If you are building out your network (or even substation, area, or zone diagram), then it is much more efficient to use the [Quick Insert Group](#). The Individual Insert group is shown below.



The six menus available on the group categorize the objects that can be placed on the oneline diagrams. A summary of what is in each menu is described below along with a picture of each of the six menus.

### Network Menu

The objects in this menu represent network model objects. Generally these represent actual physical devices such as a generator or transmission line. They are similar to the model objects found in the **Network** Folder on the Explore Pane of the [Model Explorer](#).

### Aggregation Menu

The objects in this menu represent model objects that represent an aggregation of other model objects such as interfaces or injection groups. They are similar to the data objects found in the **Aggregation** Folder on the Explore Pane of the [Model Explorer](#).

### Background Menu

The objects in this menu represent objects that are only background text and do not link to any of the objects you would look at in the [Model Explorer](#). Some of the objects link to external objects such as documents, but not to elements of the model.

### Pies/Gauges Menu

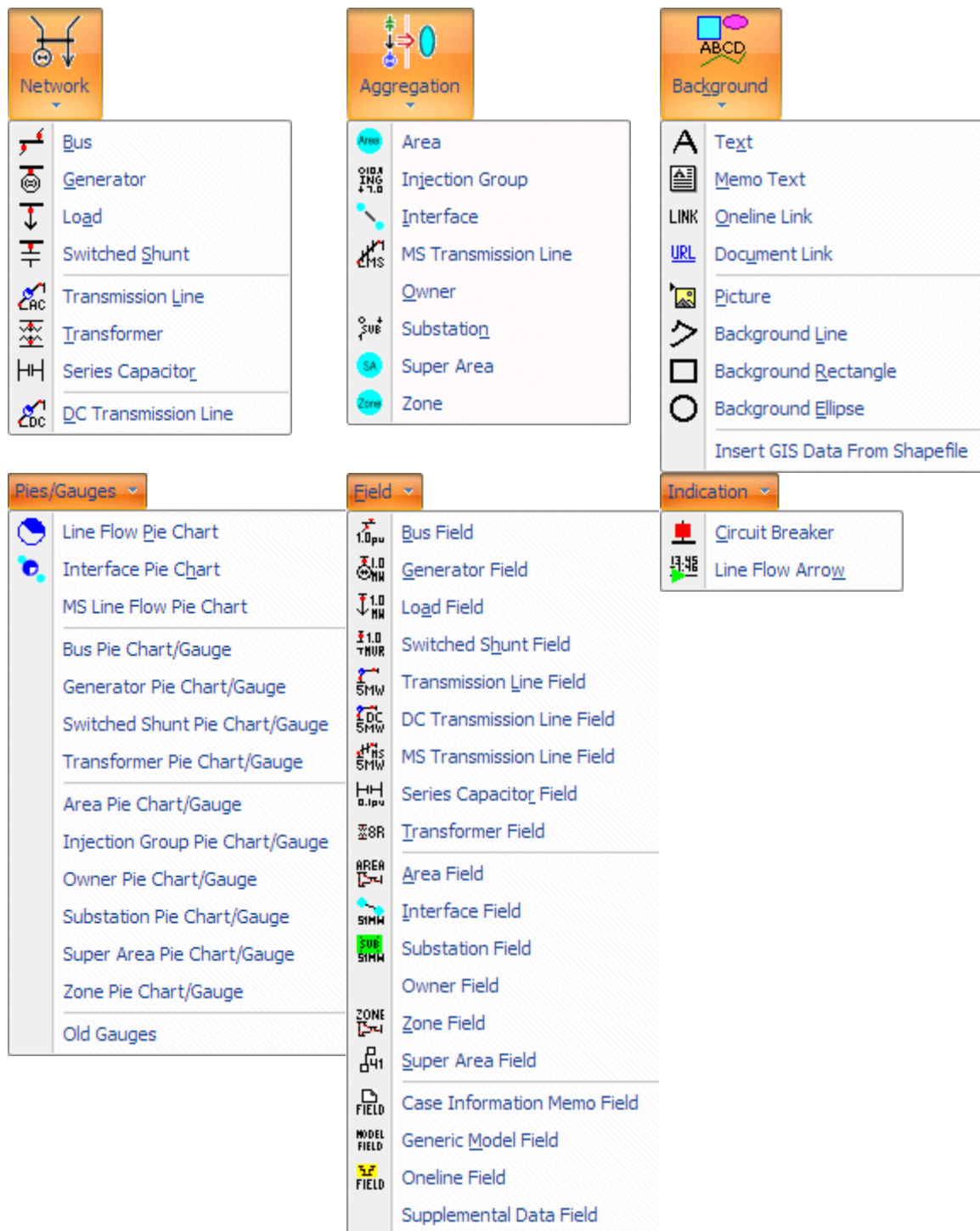
The objects in this menu represent pie charts or gauges which link to the model data found in the [Model Explorer](#).

### Field Menu

The objects in this menu represent fields which link to the model data found in the [Model Explorer](#).

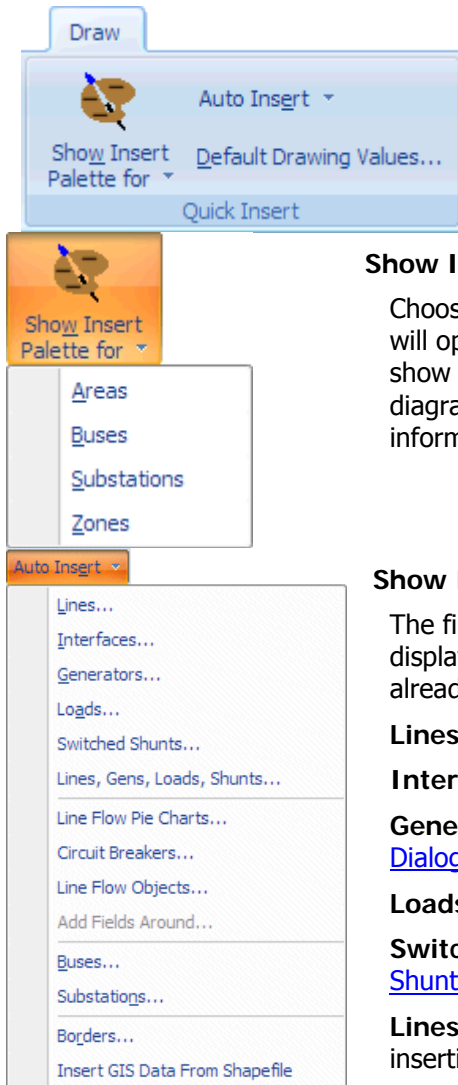
### Indication Menu

The objects in this menu represent indications of status or flow for objects that are found in the model data found in the [Model Explorer](#).



## Quick Insert Ribbon Group

The Quick Insert ribbon group contains buttons for creating a oneline diagram for a system model you have already read into Simulator. If you are building out your network (or even substation, area, or zone diagram), then using the Insert Palettes along with the Auto Insert routines is most efficient. If you would instead like help on inserting a single object on your oneline diagram see the [Individual Insert](#) ribbon group topic. Both of the ribbon groups are on the [Draw](#) ribbon tab. The Quick Insert ribbon group is shown as follows.



### Show Insert Palette for Menu

Choosing any of the items in the Show Insert Palette for Menu will open up its respective Insert Palette. This Insert palettes show a list of all objects that are already shown on the oneline diagram as well as a list of objects which are not. For more information see the [Using the Insert Palettes](#) help topic.

### Show Insert Palette for Menu

The first six options are used to automatically insert network display objects that are connected to bus objects that have already been drawn on the oneline.

**Lines** : Shows the [Automatically Insert Lines Dialog](#)

**Interfaces** : Shows the [Automatically Insert Interfaces Dialog](#)

**Generators** : Shows the [Automatically Insert Generators Dialog](#)

**Loads** : Shows the [Automatically Insert Loads Dialog](#)

**Switched Shunts** : Shows the [Automatically Insert Switched Shunts Dialog](#)

**Lines, Gens, Loads, Shunts** : Shows a dialog for auto-inserting lines, generators, loads, and shunts simultaneously.

The next four options provide methods for adding pie charts, fields, circuit breakers and line flow objects around objects that have already been drawn on the oneline diagram.

**Line Flow Pie Charts** : Select this to automatically insert [line flow pie charts](#) on the transmission lines or transformers objects that are selected.

**Circuit Breakers** : Select this to automatically insert [circuit breakers](#) on the transmission lines or transformers objects that

are selected.

**Line Flow Objects** : Select this option to automatically insert line flow objects on the transmission lines or transformers objects that are selected.

**Add Fields Around** : Select this option to insert fields around the presently selected objects. It will then bring up the [Inserting and Placing Multiple Display Objects Dialog](#)

The follow two options provide a mechanism for auto-inserting bus or substation online objects. You must have geographic longitude/latitude information in order to use these options.

**Buses** : Shows the [Automatically Insert Buses Dialog](#)

**Substations** : Shows the [Automatically Insert Substations Dialog](#)

The last two options provide a mechanism for inserting background lines that represent geographic borders or from a Shapefile.

**Borders** : Shows the [Automatically Insert Borders Dialog](#)

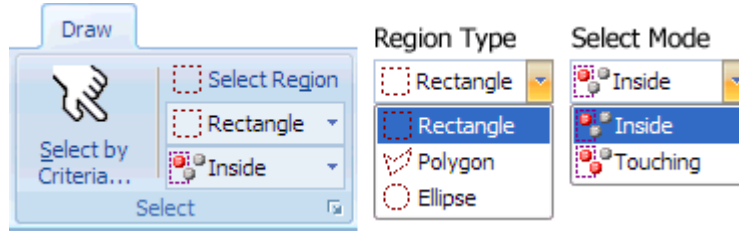
**Insert GIS Data From Shapefile** : Shows the [Shapefile Import Dialog](#)

---

## Select Ribbon Group

---

The Select ribbon group contains options to help select multiple oneline objects simultaneously. The features in this topic are only available when you are in [Edit Mode](#). This ribbon group is on the [Draw](#) ribbon tab. The Group is shown below.



The features available on this ribbon group are described below.



### Select by Criteria

Click on the Select by Criteria button to open the [Select by Criteria Dialog](#). On oneline diagrams of even a modest size, this is an extremely useful feature and PowerWorld highly encourages you to use this feature.

### Select Region

Use to select all objects in a particular region of the oneline. When the **Region Type** is **Rectangle** or **Ellipse**, then after clicking the button, click and hold the left mouse button on the oneline at the point where you would like to begin the selection. Then, drag the mouse to size the selection shape. Finally, let go of the mouse button once every object you need to select has been selected by the selection rectangle. When the Region Type is Polygon, then after clicking this button single click on your oneline diagram for each vertex of the polygon. Finally, finish the selection by double-clicking the mouse after which all objects inside your polygon will be selected.

Depending on the **Select Mode** below, every object located entirely **Inside** the region, or if you have chosen **Touching**, then every object inside or partially touched will be selected. Selected objects will have handles appear indicating that they have been selected.

### Region Type

Specify a **Rectangle**, **Polygon**, or **Ellipse**. This controls the shape used to select objects on the oneline diagram when using the **Select Region** button above.

### Select Mode

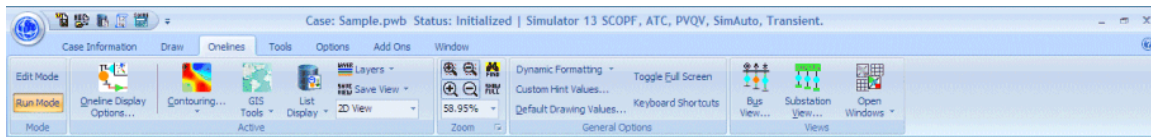
This combo box specifies how the **Select Region** button above works. Choose **Inside** in order to select only the objects completely inside the selection rectangle. Choose **Touching** in order to select every object either partially touched by or completely inside the selection rectangle.

There are also other ways to select multiple objects

- You can also select multiple objects manually by holding down the SHIFT key which clicking on objects in Edit Mode. This will also add to your selection.
- If you hold down the SHIFT and CTRL keys simultaneously and then click on your oneline diagram and select a region it has the same affect as click on the **Select Region** button and selecting a region on your oneline

## Onelines Ribbon Tab

The Onelines ribbon tab generally consists of buttons and menus that provide access to modifying the appearance of a oneline diagram you have already created. For access to features regarding adding, moving, resizing, and formatting objects, see the Draw ribbon tab. The Oneline ribbon is shown below.



The Options ribbon consists of buttons and menus that provide access to customizing the software options. The ribbon is shown below.

### Mode Ribbon Group

#### Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

#### Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

### Active Ribbon Group

The Active ribbon group contains basic methods for customizing the appearance of basic analysis tools for customizing an existing oneline diagram. For more detailed help see the [Active](#) ribbon group topic

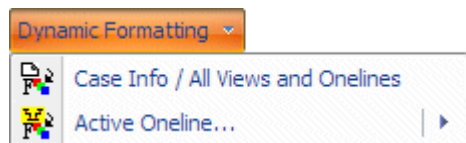
### Zoom Ribbon Group

The Zoom ribbon group contains buttons for navigating a oneline diagram. For more detailed help see the [Zoom](#) ribbon group topic

### General Options Ribbon Group

#### Dynamic Formatting Menu

This menu provides access to setting the [Dynamic Formatting Options](#) that apply to the active oneline, as well as formatting that applies to all onelines and Case Information Displays.



#### Custom Hint Values

Provides access to setting the [Custom Hint Values](#) that apply to all oneline objects..

#### Default Drawing Values

Provides access to the [Default Drawing Values](#).

#### Toggle Full Screen

Click this button to cause the presently active oneline to be toggled to full screen.

**Keyboard Shortcuts**

Click on this button to open the dialog for defining [Keyboard Shortcuts](#) which allow you to open a oneline or auxiliary file directly from a function key.

**Views Ribbon Group****Bus View**

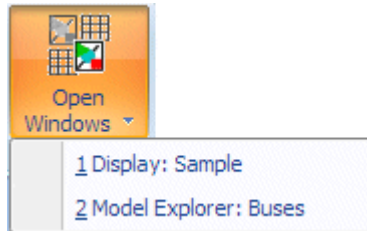
Opens the [Bus View Online](#) which provides a graphical display which allows you to quickly browse information about a bus and everything connect to that bus.

**Substation View**

Opens the [Substation View Online](#) which provides a graphical display which allows you to quickly browse information about a bus and everything connect to that substation.

**Open Windows Menu**

The Open Windows Menu provides a convenient method of switching between open windows. This includes User Interface Dialogs, Case Information Displays, and Oneline diagrams.

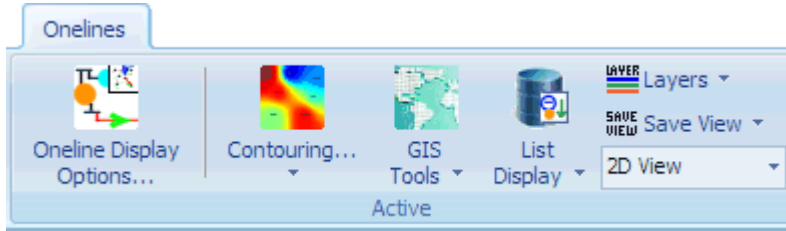


---

## Active Ribbon Group

---

The Active ribbon group contains basic methods for customizing the appearance of basic analysis tools for customizing an existing online diagram.



### Online Display Options

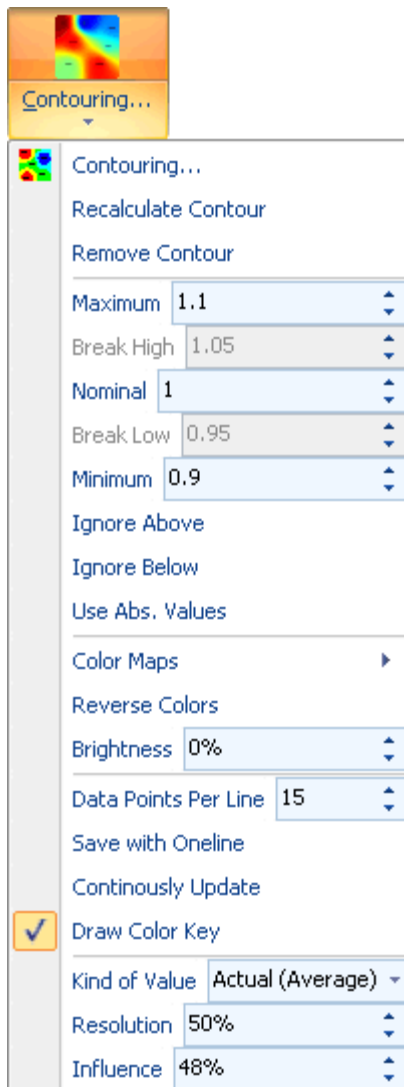
Opens the [Online Display Options Dialog](#).

### Contouring and Menu

Click the top part of the button to open the [Contouring Dialog](#).

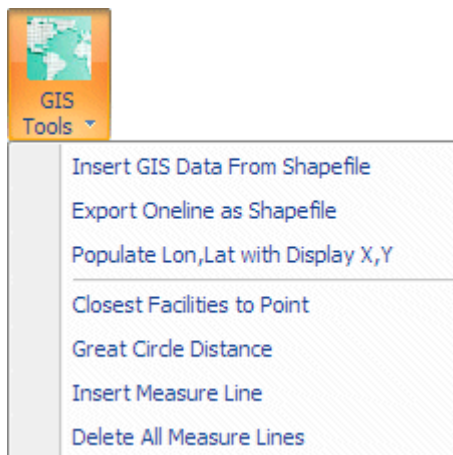
Otherwise, click the drop-down to provide direct access to many of the Contouring Options.





### GIS Tools Menu

Provides access to many Geographic Information System related tools that have been built into the software.



**Insert GIS Data From Shapefile** : Opens the [Shapefile Import dialog](#).

**Export Online as Shapefile** : Choose this to [Export the Online as a Shapefile](#).

**Populate Lon,Lat with Display X,Y** : Choose this to populate the Bus records latitude and longitude information based on the locations on the present oneline diagram. For more information see the [Populate help topic](#).

**Closest Facilities to Point** : Choose this to bring up the [Closest Facility to Point feature](#).

**Great Circle Distance** : Choose this to calculate the [Great Circle Distance between two points](#).

**Insert Measure Line** : Provides access to the [Insert Measure Line Feature](#).

**Delete All Measure Lines** : Provides access to the [Delete All Measure Lines feature](#).

### List Displays Menu

Contains access to several ways of viewing the oneline objects inside of "[case information displays](#)". Of course instead of seeing case information, you instead see information about the oneline objects.



**All Display Objects:** Opens the [Display Explorer](#) providing access to all objects on the active oneline.

**Unlinked Display Objects** : Opens the [Unlinked Display Objects Dialog](#).

**Only Selected Display Objects** : Opens the [Display Explorer](#) providing access to objects on the active oneline, but defaults that explorer to only show the presently selected objects.

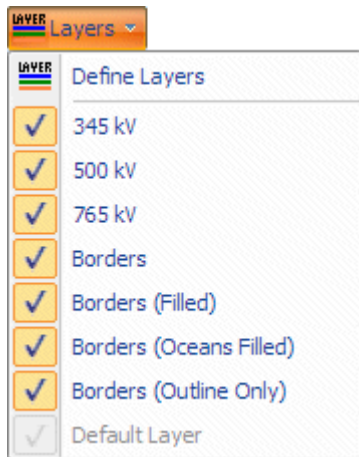
**Browse Open Onelines:** Brings up the [Browse Open Onelines dialog](#).

**Display Objects Export Format Description** : Opens a [Display Objects Export Format Description](#) dialog.

### Layers Menu

The Layers Menus provide access for opening the [Define Layers Dialog](#) for the presently active oneline diagram.

Underneath the button for defining layers is a list of the presently defined layers. You may check or uncheck the various layers to hide or show the objects in those layers.

**Save View**

Choose this option to open the [Save Views Dialog](#).

**2D View / 3D View**

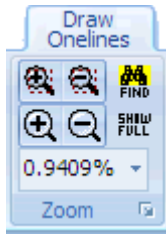
Toggle between 2D and 3D to change the presently active online between the 2D and 3D modes.

---

## Zoom Ribbon Group

---

The Zoom Ribbon Group is on both the [Onelines](#) ribbon tab and the [Draw](#) ribbon tab. To display large detailed power systems, Simulator's onelines possess zooming and panning capabilities. The Zoom Ribbon Group enables you to prescribe a zoom level either by directly specifying a zoom value or by selecting a rectangular region of the diagram on which to focus. There is also a button to open a dialog box from which you can select a bus on which to center the display. See [Zooming and Panning](#) for more information.



This toolbar contains the following several buttons for zooming the presently active oneline.

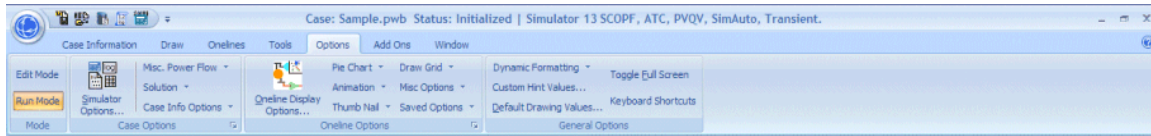
- **Zoom Area In /Out** : The top two buttons can be clicked to zoom or out on a particular area of the oneline. When you click these buttons, the cursor will change to a cross-hair waiting for you to choose the region to zoom into (or out of)
- **Zoom In / Out** : The middle two buttons can be clicked to change your mouse into a zooming tool. After choosing the button with a plus sign, each time you click on the oneline you will zoom in on that location. When choosing the button with a negative sign you will zoom out instead.
- **Present Zoom Level**: the number represents the present zoom level and can be manually changed to modify the zoom level.
- **Show Full** : Automatically zooms and pans the presently active oneline diagram so that all objects on the oneline diagram can be seen.
- **Find**: Provide access to the [Find Object on Oneline Dialog](#)

---

## Options Ribbon Tab

---

The Options ribbon consists of buttons and menus that provide access to customizing the software options. All of the buttons on this ribbon are also available on one of the other ribbons, however this ribbon brings all the options in the software into one place. The ribbon is shown below.



The Options ribbon is broken into four ribbon groups

### Mode Ribbon Group

#### Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

#### Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

### Case Options Ribbon Group

#### Simulator Options

Opens the [Simulator Options Dialog](#).

#### Misc. Power Flow Menu

Provides more direct access to some miscellaneous options that are also found on the [Simulator Options Dialog](#).

#### Solution Menu

Provides more direct access to some options related to the power flow solution algorithm. These options are all also available on the [Simulator Options Dialog](#).

#### Case Info Options Menu

Provides more direct access to some options related to the appearance of case information displays. These options are all also available on the [Case Information Display Options](#) on the [Simulator Options Dialog](#).

### Online Options Ribbon Group

#### Online Display Options

Opens the [Online Display Options Dialog](#).

#### Pie Chart Menu

Provides more direct access to options related to Pie Charts. These options are all also available on the [Online Display Options Dialog](#) on the [Pie Charts/Gauges](#) page.

#### Animation Menu

Provides more direct access to options related to Animated Arrows. These options are all also available on the [Online Display Options Dialog](#) on the [Animated Flows](#) page.

#### Thumb Nail Menu

Provides more direct access to options related to Thumb Nail View. These options are all also available on the [Online Display Options Dialog](#) on the [ThumbNail View](#) page.

#### **Draw Grid Menu**

Provides more direct access to options related to the Draw Grid. These options are all also available on the [Online Display Options Dialog](#).

#### **Misc. Options Menu**

Provides more direct access to miscellaneous options related to oneline diagrams. These options are all also available on the [Online Display Options Dialog](#).

#### **Saved Options Menu**

This menu provides direct access to the for toggling between different sets of options. These different sets of options are defined on the [Online Display Options Dialog](#) by clicking on the **Save Options to Case** button on that dialog..

### **General Options Ribbon Group**

#### **Dynamic Formatting Menu**

This menu provides access to setting the [Dynamic Formatting Options](#) that apply to the active oneline, as well as formatting that applies to all onelines and Case Information Displays.\_

#### **Custom Hint Values**

Provides access to setting the [Custom Hint Values](#) that apply to all oneline objects..

#### **Default Drawing Values**

Provides access to the [Default Drawing Values](#).

#### **Toggle Full Screen**

Click this button to cause the presently active oneline to be toggled to full screen.

#### **Keyboard Shortcuts**

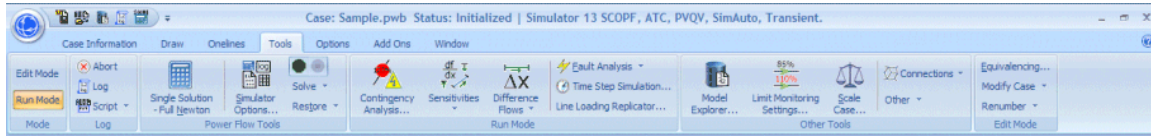
Click on this button to open the dialog for defining [Keyboard Shortcuts](#) which allow you to open a oneline or auxiliary file directly from a function key.

---

## Tools Ribbon Tab

---

The Tools ribbon consists of buttons and menus that provide access to the analysis tools that come with all versions of PowerWorld Simulator. Note: Features related to the various add-on tools including OPF, SCOPF, PVQV Curve, and ATC are found on the [Add Ons](#) ribbon tab. The tools ribbon is shown below



The Tools ribbon is broken into six ribbon groups

### Mode Ribbon Group

#### Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

#### Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

### Log Ribbon Group

#### Abort

Terminates the current Power Flow Solution. If the application is performing a timed simulation, pressing the abort button will pause the simulation. See [PowerWorld Simulation Control](#) for more details.

#### Log

Toggles the display of the [message log](#) window. The log window shows what is going on with the Power Flow Solution process and may prove useful when you are trying to track down a problem with a non-converging model.

#### Script

Opens the Script dialog, which can be used to call [script commands](#) or open [auxiliary files](#) containing script commands and data modifications. Note that the drop-down next to the Script button give convenient access to the [Quick Auxiliary Files](#).

### Power Flow Tools Ribbon Group

This group represents the basic solution actions for solving the power flow.

For more detailed help see the [Power Flow Tools](#) ribbon group topic.

### Run Mode Ribbon Group

The Edit Mode Ribbon Group contains basic analysis tools that are only available in [Run Mode](#).

For more detailed help see the [Run Mode](#) ribbon group topic.

### Other Tools Ribbon Group

The Edit Mode Ribbon Group contains basic analysis tools that are normally available in both [Run Mode](#), and [Edit Mode](#).

For more detailed help see the [Other Tools](#) ribbon group topic.

### **Edit Mode Ribbon Group**

The Edit Mode ribbon group contains basic analysis tools that are only available in [Edit Mode](#).

For more detailed help see the [Edit Mode](#) ribbon group topic.

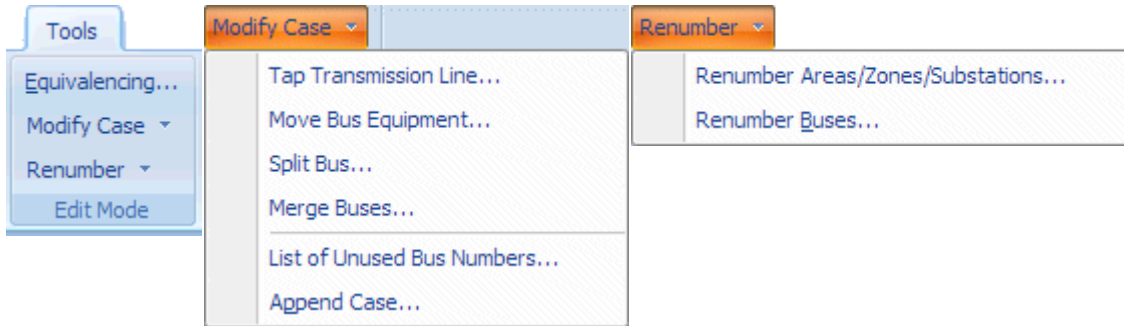


---

## Edit Mode Ribbon Group

---

The Edit Mode ribbon group contains basic analysis tools that are only available in [Edit Mode](#). The Group is shown below.



### Equivalencing

Click this button to open the [Equivalencing Dialog](#).

### Modify Case Menu

The Modify Case Menu contains actions related to modifying the network model.

**Tap Transmission Line:** Opens the [Tap Transmission Line Dialog](#).

**Move Bus Equipment:** Opens the [Move Bus Equipment Dialog](#).

**Split Bus:** Open the [Split Bus Dialog](#).

**Merge Buses:** Opens the [Merge Buses Dialog](#).

**List of Unused Bus Numbers:** Clicking this button to get a [list of unused bus numbers](#) in the model.

**Append Case:** Provides access to [opening a case while appending](#).

### Renumber Menu

The Renumber Menu contains actions related to renumbering data in the model.

**Renumber Areas/Zones/Substations:** Opens the [Renumber Areas, Zones and Substations Dialog](#).

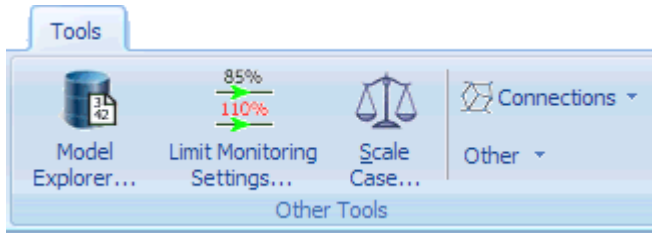
**Renumber Buses:** Opens the [Renumber Buses Dialog](#).

---

## Other Tools Ribbon Group

---

The Other Tools ribbon group contains basic analysis tools that are normally available in both [Run Mode](#) and [Edit Mode](#). The group is shown below.



### Model Explorer

Opens the [Model Explorer](#) giving access to all the [case information displays](#) for the model.

### Limit Monitoring Settings

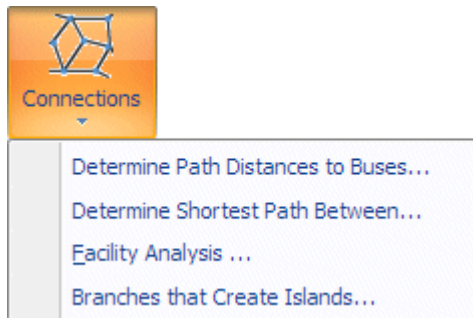
Opens the [Limit Monitoring Settings Dialog](#).

### Scale Case

Opens the [System Scaling Dialog](#).

### Connections Menu

The Connection Menus contains actions related to analyzing the interconnectedness of the network model. These may be considered Graph Theory routines.



**Determine Path Distances to Buses:** Opens the [Determine Patch Distances from Buses or Group Dialog](#).

**Determine Shortest Path Between:** Opens the [Determine Shortest Path Between Dialog](#).

**Facility Analysis:** Open the [Facility Analysis Dialog](#).

**Branches that Create Islands:** Opens the [Branches that Create Islands Dialog](#).

### Other Menu

The Other Menu contains other miscellaneous features.



**Governor Power Flow:** Opens the [Governor Power Flow Dialog](#).

**Set Generator Part. Factors:** Opens the [Set Generator Participation Factors Dialog](#)

**Set Selected Field for Network Cut:** Opens the [Set Selected Field for Network Cut Dialog](#)

**Set Selected From Selection:** When a selection is chosen in [Edit Mode](#), click this to set the Selected? field to YES for that selection.

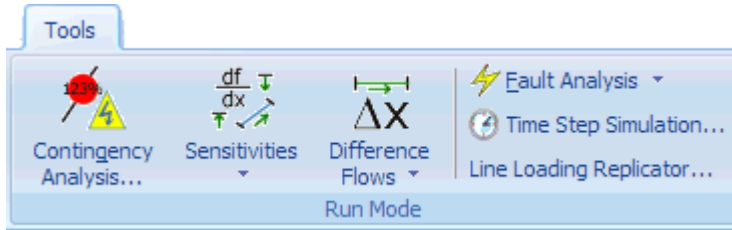
**Charts Menu:** The Charts menu is only available in [Run Mode](#). It provides access to various charts including the [Area ACE Chart](#), [Area Load and Generation Chart](#), [Area Losses Chart](#), [Area MW Transactions Chart](#), and [Area Average Cost Chart](#).

---

**Run Mode Ribbon Group**


---

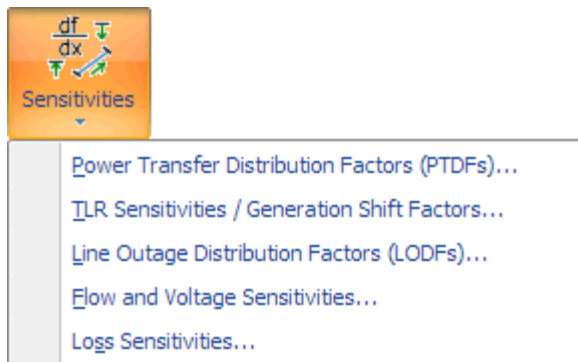
The Run Mode ribbon group contains basic analysis tools that are only available in [Run Mode](#). The group is shown below.


**Contingency Analysis**

Click this button open the [Contingency Analysis Dialog](#) providing access to the automated processing of a list of contingencies.

**Sensitivities Menu**

The Sensitivities Menu contains a list that provides access to all the sensitivity calculations available.



[Power Transfer Distribution Factors \(PTDFs\)](#) opens its associated dialog.

[TLR Sensitivities / Generation Shift Factors](#) opens its associated dialog.

[Flow and Voltage Sensitivities](#) opens its associated dialog.

[Loss Sensitivities](#) opens its associated dialog.

**Difference Flows Menu**



Difference Flows: Opens the [Difference Flows Dialog](#)

Present Topological Differences From Base Case: Opens the [Present Topological Differences From Base Case Dialog](#).

The remaining options in this menu all replicate the same behavior available on the [Difference Flows Dialog](#). For more details so that dialog.

### **Fault Analysis**

Click this button to open the [Fault Analysis Dialog](#).

### **Time Step Simulation**

Click this button to open the [Time Step Simulation Dialog](#).

### **Line Loading Replicator**

Click this button to open the [Line Loading Replicator Dialog](#).

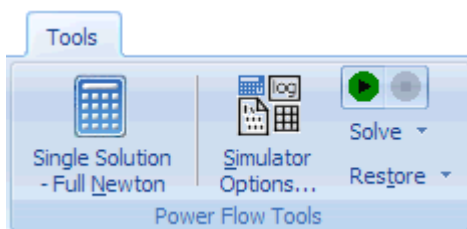
---

## Power Flow Tools Ribbon Group

---

The main function of the PowerWorld Simulator is to simulate the operation of an interconnected power system. The simulation is accomplished by selecting **Single Solution – Full Newton** from either the [Quick Access Toolbar](#) or the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab. This activity performs a single Power Flow Solution. See [Solving the Power Flow](#) for more information.

The following tasks are also available from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab.



### Single Solution – Full Newton

The main function of the PowerWorld Simulator is to simulate the operation of an interconnected power system. The simulation is accomplished by selecting **Single Solution – Full Newton** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab. This activity performs a single Power Flow Solution. See [Solving the Power Flow](#) for more information. Also note that if presently in Edit Mode, you will be automatically taken to Run Mode when choosing this.

### Simulator Options

When choosing any of the solution methods, the options as specified in the [Simulator Options](#) will affect what calculation is performed. solves the power flow using the either the Full Newton AC load flow or the DC Approximation load flow, as specified in the [Simulator Options](#).

### Play Button

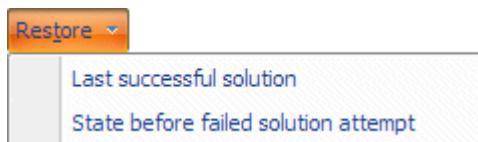
Click this button to cause repeated solutions of the system and to cause the animation of oneline diagrams

### Pause/Stop Button

Click this button to stop the action of the Play button.

### Restore Menu

The Restore menu offers option to either Restoring a Previous Solution or a Restore a Previous State



Sometimes a power flow attempt won't converge to a solution. When this occurs, the voltages and angles calculated by the solution engine will not satisfy the real and reactive power balance constraints at each bus. Then, the state currently stored in memory will not be an actual system operating point. It is often very difficult to coax the system to solve once it has failed to converge.

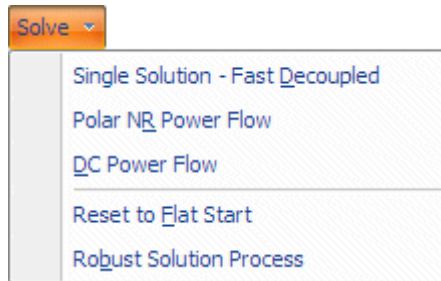
To help you recover from a solution attempt that has failed to converge (both timed simulations and single solutions), Simulator offers you two options.

After Simulator solves a system successfully, it will store the voltages and angles it found in memory. If the changes that you then make to the system result in a system that can't be solved, you can select **Restore > Last Successful Solution** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab to reload the results of the last converged solution. After reloading this information, Simulator will re-solve the system and refresh all displays.

In addition to restoring the last converged solution, Simulator also gives you the ability to restore the state of the system as it was just prior to the unsuccessful solution attempt. This can be thought of as "un-doing" the effect of the solution attempt. Before attempting a solution, Simulator stores the state of the system in memory. If it solves the power flow successful, Simulator will discard this pre-solution state. However, if the power flow fails to converge, Simulator will keep the state in memory. To recover it, select **Restore > State Before Solution Attempt** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab. Simulator will replace the non-converged post-solution state with the pre-solution state and refresh the displays. You can then play with the system to try to make it easier to solve.

If you are working with large systems, you should be aware that saving these system states can consume a lot of memory. Therefore, Simulator offers you the option to disable one of both of these features. To do this, select **Simulator Options** from the **Case Options** ribbon group on the [Options](#) ribbon tab to open the [Simulator Options Dialog](#), and then go to the Storage sub-tab of the Solution tab. You will see two checkboxes there that can be modified to control whether or not these extra system states are saved.

### Solve Menu



### Single Solution – Full Newton

The main function of the PowerWorld Simulator is to simulate the operation of an interconnected power system. The simulation is accomplished by selecting **Single Solution – Full Newton** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab, or from the [Quick Access Toolbar](#). This activity performs a single Power Flow Solution. See [Solving the Power Flow](#) for more information. Also note that if presently in Edit Mode, you will be automatically taken to Run Mode when choosing this.

### Polar NR Power Flow

This perform a Newton Rhapsod power flow solution using polar coordinates instead of rectangular coordinates as is normally done in Simulator.

### DC Power Flow

This automatically change the Simulator Options to use the [DC Approximation](#) and then immediately performs the DC power flow solution.

### **Reset to Flat Start**

Select **Solve > Reset to Flat Start** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab to initialize the Power Flow Solution to a "flat start." A flat start sets all the voltage magnitudes and generator setpoint voltages to 1.0 per unit and all the voltage angles to zero. Usually, a flat start should be used only if the power flow is having problems converging. You can also use the flat start option on the [Simulator Options Dialog](#) to initialize every solution from a flat start.

### **Robust Solution Process**

The Robust Solution Process provides a method to attempt to reach a solution when the standard load flow (Newton-Raphson) solution fails. The robust process performs a solution in a series of steps.

First, the robust solution will turn off all controls in the case. Then the load flow will be solved using a fast decoupled power flow. If the fast decoupled power flow reaches a solution, Simulator then immediately solves the load flow using the Newton-Raphson load flow, still keeping the controls turned off. If the Newton-Raphson solution is also successful, Simulator will begin adding controls back into the solution process, one type of control at a time. Thus the generator MVAR controls are added back in, and the load flow is resolved. Then the switched shunt controls are restored, and the load flow is again resolved. Simulator will continue in this manner by reintroducing next the LTC control, followed by the area interchange control, and lastly the phase shifter control. Furthermore, when reintroducing the phase shifter control, the controls are added one at a time for each phase shifter, with a load flow solution occurring after each.



## Window Ribbon Tab

The Window ribbon consists of buttons and menus that provide access to customizing the windows in the user interface. It also provides access to help topics and topics regarding the Auxiliary File Format. The ribbon is shown below.



The Case Information ribbon is broken into four ribbon groups

### Mode Ribbon Group

#### Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

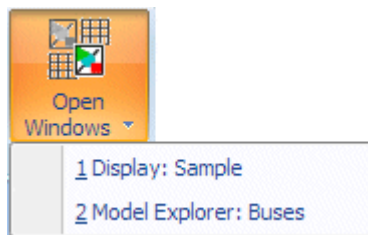
#### Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

### Window Ribbon Group

#### Open Windows Menu

The Open Windows Menu provides a convenient method of switching between open windows. This includes User Interface Dialogs, Case Information Displays, and Online diagrams.



#### Switch to Free-Floating Windows

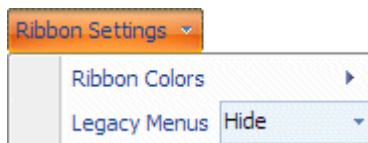
Choosing this option toggles Simulator from containing all diagrams within the Simulator program shell to all windows related to Simulator being free-floating on your display.

#### Refresh Displays

Redraws (refreshes) each of the open windows. Simulator usually automatically refreshes the open windows as necessary. However, this action allows you to trigger the refresh when you want it.

#### Ribbon Settings Menu

This menu contains options for customizing your ribbon.



**Ribbon Colors** : You can change the color scheme to either **Blue**, **Silver** or **Black**

**Legacy Menus** : Choose to Hide or Show a special Menus ribbon tab. The Menus ribbon tab mimics the menu and toolbar structure that existed in older versions of Simulator.

#### **Tile**

Rearranges the open online diagrams such that the total window area is divided equally among all of them and each is completely visible.

#### **Cascade**

Rearrange all open windows such that all they appear on top of each other while leaving the title bars visible.

#### **Toggle Full Screen**

Toggle full screen makes the currently selected diagram window switch to full screen mode. Full screen mode will dedicate the screen to the window, with all other windows, including the Simulator menus and toolbars, to be hidden. To get back to normal mode, right-click on the diagram in full screen mode and select Toggle Full Screen from the popup menu.

### **Help Ribbon Group**

#### **Contents**

Opens the [Simulator Options Dialog](#).

#### **Set Help File**

Opens the [Case Description Dialog](#).

#### **About**

Click this to open the About Dialog. The About dialog contains information about the software as well as showing you the Build Data of your version of Simulator. PowerWorld release patch version of Simulator on our website. These patch versions are named by date and is shown on the About Dialog.

#### **PowerWorld Website**

Click this to open your default web browser to the PowerWorld Corporation Website.

#### **Check for Updates**

This button will open the PowerWorld website to the patch download page. You will be required to enter your Account Username and Product ID (password) to check for and download patches for your version of Simulator.

### **Auxiliary File Ribbon Group**

#### **Load Auxiliary**

Choose this option to load an [Auxiliary File](#).

#### **Load Display Auxiliary File**

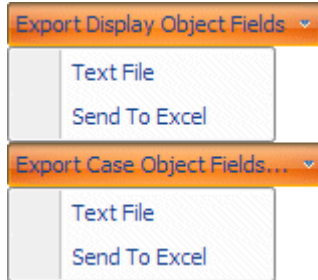
Choose this option to load a [Display Auxiliary File](#) to append to the active online.

#### **Auxiliary File Format**

Opens the [Bus View Online](#) which provides a graphical display which allows you to quickly browse information about a bus and everything connect to that bus.

#### **Export Display Object Fields Menu and Export Case Object Fields Menu**

Choose these options to export a list of fields for each type of object in a case (or a Oneline Display). The list also indicates which fields are [key fields](#) and [required fields](#) for each object. You can output this list of fields as either a text file or into Excel.



---

## **Chapter 2: Using Oneline Diagrams**

---

Simulator has been designed to be both highly graphical and interactive. The key to making effective use of Simulator lies in understanding the oneline diagrams. This chapter provides essential information on how to use and customize onelines.

The following material is included:

- Oneline Diagram Overview
- Oneline Tools and Options
- Printing Oneline Diagrams
- Relationship Between Display Objects and the Power System Model
- GIS Tools

---

## Online Diagram Overview

---

The purpose of the online diagram is to show information about the power system graphically. Such displays are called online diagrams (onelines) because the actual three-phase power system components are represented using a single line. Simulator onelines "come alive" via:

- [Animation](#)
- [Contouring](#)
- [Zooming and panning](#) capability and
- [Conditional display of objects](#)

Additionally, a key aspect of Simulator is the ease with which it allows the user to examine and modify many of the shown on the online diagram.

The user may open any number of online diagrams, including multiple copies of the same online.

---

## Relationship Between Display Objects and the Power System Model

---

### Display Object:

An item shown on a oneline diagram. Display objects typically have an associated model object. Examples include buses, transmission lines, transformers, generators and loads. Display objects not associated with a model object are called unlinked objects.

### Model Object:

A power system element contained in a case.

A key strength of the Simulator is its ability to allow users to manipulate a power system model graphically. This capability greatly simplifies the work involved in developing or maintaining a power system case for both novice and advanced users. However, it is important to keep in mind the distinction between the shown on the onelines and the actual power system model, consisting of model objects. A key concept is that any number of display objects, including none at all, can be associated with a single model element.

Simulator uses a bus-oriented model. In other words, the model objects are either the buses themselves, objects that are radially attached to a bus (i.e., loads, generators and switched shunts), or objects that join two buses (i.e., transmission lines, transformers or dc lines). As long as there is a one-to-one mapping between display objects and model objects, the distinction between the two could be made entirely transparent to the user.

It is reasonable, and often quite useful, to use more than one display object to represent a single model object. For example, by using the [Conditional display of objects](#) feature, two bus display objects could be used on a single oneline to represent the same bus. One bus might be visible over a particular zoom range, while another, with perhaps a different size/thickness, is visible over another range. Alternatively, the same bus could be represented using display objects drawn on separate onelines.

An ambiguity arises when the user uses the Cut command to delete an object. Is he or she deleting just the Display Object or both the Display Object and the Model Object? To alleviate the problem, Simulator prompts you when you are deleting a display object with an associated model object to delete both the Display Object and its associated model object record, delete just the display object, or cancel the delete.

In addition, there is no requirement that model objects have a corresponding display object. Thus, you could use the oneline diagram to show just a fraction of the total system buses and other devices. You can use the [Case Information](#) menu to view the model objects directly regardless of whether or not they are shown on a oneline.

---

## Online Local Menu

---

The local menu provides access to a number of options and tools directly from the online. To display the local menu, position the cursor on an empty portion of the online then click the right mouse button. While most options are always available on the popup menu, a few options are only available on the popup menu in run mode.

### Find Object on Online (Edit and Run Modes)

Displays the Find Object on Online tab of the [Zoom, Pan, and Find Objects](#) dialog.

### Online Display Options (Edit and Run Modes)

Displays the [Online Display Options](#) dialog. This dialog allows you to customize the appearance of the online.

### Pan/Zoom Control (Edit and Run Modes)

Displays the Zoom/Pan tab of the [Zoom, Pan, and Find Objects](#) dialog.

### Area Information Dialog (Edit and Run Modes)

Displays the [Area Information Dialog](#) for the bus nearest the cursor when the local menu was opened

### Contouring (Run Mode Only)

Displays the [Contour Options Dialog](#). This dialog allows you to contour the system voltage magnitudes or angles.

### Difference Flows (Run Mode Only)

Displays the [Difference Flows Dialog](#). This dialog is used to compare two power system operating points.

### Toggle Flow Visualization (Run Mode Only)

Switches the online's animated flows between displaying actual flows and [power transfer distribution factors](#).

### Dynamic Formatting (Active Online) (Edit and Run Modes in Online Diagrams)

Displays the [Dynamic Formatting Dialog](#). This dialog is used to change the rendering of objects in online diagrams according to the state of the represented object in the power system.

### Dynamic Formatting (All Views) (Edit and Run Modes in Bus and Substation Views)

Displays the [Dynamic Formatting Dialog](#). This dialog is used to change the rendering of objects in bus and substation views according to the state of the represented object in the power system.

### All Display Objects (Edit and Run Modes)

Opens the [Display Object case information display](#), showing all the objects contained in the online diagram.

### Only Selected Display Objects (Edit Mode Only)

Opens the [Display Object case information display](#), showing only the currently selected objects in the online diagram, by automatically checking the **Show Only Objects Selected on Display** box.

### Load Display Auxiliary File (Edit and Run Modes)

Loads a [display auxiliary file](#).

**Edit Screen Layers (Edit and Run Modes)**

Opens the [Screen Layers case information display](#).

**Show Layer (Edit and Run Modes)**

Allows activation of any saved screen layer.

**Print Oneline (Edit and Run Modes)**

Sends a copy of the oneline diagram to the printer. Selecting this option has the same affect as selecting **File, Print Oneline** from the main menu. See [Printing Oneline Diagrams](#) for more information.

**Copy Image to Clipboard (Edit and Run Modes)**

Copies the oneline file into the Windows clipboard using the Windows Metafile format (\*.wmf). You can then paste the oneline into another program, such as a word processor. See [Copying Onelines to Other Programs](#) for details.

**Export Image to File (Edit and Run Modes)**

Saves a copy of the entire oneline diagram image to a file. Oneline image files can be saved in bitmap (\*.BMP), Windows Metafile (\*.WMF, .EMF), or JPEG format (.jpg).

**Toggle Full Screen (Run Mode Only)**

Sets the oneline diagram display window to full screen size. This option hides all toolbars and status bars. To return the window to normal, right-click on the oneline diagram and uncheck the Full Screen option.

**Embed a Display (Run Mode Only)**

Clicking on this option allows you to open another oneline diagram (or the same one) and embed the new display inside the existing display. You can choose what percentage size the embedded display should be, and where within the existing window the embedded window should be placed.

**Save/Edit/Delete View (Edit and Run Modes)**

Displays the [Save View Level Dialog](#). This dialog is used to set defined locations on the oneline for recalling specific views from a list of saved views.

**Go To View (Edit and Run Modes)**

Allows the user to go to a specific location and zoom level on the oneline by choosing from a list of saved views. This option does nothing if no views are saved.

**Form Control > (Edit and Run Modes)**

Options on this submenu allow you to resize the window, shift the window, or close the window.



---

## Online Display Options Dialog

---

The Online Display Options dialog allows you to customize the display of the online diagram. You can access this display by right-clicking anywhere on an empty portion of a online and selecting **Online Display Options** from the resulting [online local menu](#).

This dialog can also be display by going to the [Options](#) ribbon tab and choosing **Online Display Options** from the **Online Options** ribbon group.

This dialog houses several tabbed pages that govern various aspects of the online display. See [Animated Flows](#), [Display Object Options](#), [Display Options](#), [Grid/Highlight Unlinked Objects](#), [Geography/Coordinates](#), [Memo](#), [Movie](#), [Pie Charts / Gauges](#), [Thumbnail View](#), or [Substations](#) for information on the respective tabs.

### Save Options to Case

Of important note is the general option at the bottom of this dialog labeled **Save Options to Case**. What this allows you to do is define a set of online options in the dialog, and then save that definition of options *in the case* by giving the definition a name. Once you have named that set of options, you can recall it later by coming back to this dialog and choosing the name from the list of option set names that appears near the bottom of the dialog. You can define as many different sets of custom options as you wish. You can also use an option set saved with the case when you create a online view using the [Save Views](#) dialog.

Also note that once you have defined these sets of Saved Online Options, you can then quickly apply them to the active online diagram by using the **Saved Options Menu** on the [Options](#) ribbon tab.

---

## Display Options

---

### Display Detail

The Display Detail box allows you to control how much information is shown on the oneline display. This option is available only in run mode. All elements on a oneline diagram are automatically visible in edit mode. There are four choices for run mode detail filtering:

**Minimal** Show the oneline background, branch circuit breakers, generator MW output, and load MW/MVR.

**Moderate** Show all Minimal information, along with bus voltages and all line flow pie charts.

**Complete** Show all information.

**Custom** Selecting this option and then clicking the **Set...** button opens the Custom Detail Dialog, which allows you to customize the oneline diagram to hide objects that do not meet your desired specifications. This dialog looks and works very similar to the [Select By Criteria](#) tool, allowing you to choose objects filtered by area, zone, voltage, and screen layer.

Of course, in order for a certain display object to appear, it must have been placed there by the person who designed the oneline.

### Emphasize Specific Objects

This option allows you to choose to "emphasize" desired elements on a oneline diagram. The emphasis is in the form of the emphasized elements being in full color, with de-emphasized elements being muted colors.

To choose emphasis of certain elements, first check the box labeled **Do Emphasis**. When you first check this box, it will open the Emphasis Filter dialog, which looks and works very similar to the [Select By Criteria](#) tool. You can choose the oneline elements you wish to emphasize, and include filtering by area, zone, voltage, and screen layer.

The degree of emphasis (muting of de-emphasized elements) can be controlled by the **Emphasis Amount** slide bar.

### Background Color

Select **Change Background Color** to select a different color for the oneline diagram background color. Select the **Set as Default Background Color** option to set the background color as the default for all oneline diagrams.

### Use Absolute Values for MW Line Flows

If checked, this option will cause all MW flow text fields for lines to be displayed as the absolute value of the flow. Otherwise, the MW flow text fields will be positive near the source end of the line and negative near the sink end of the line.

### Use Absolute Values for Mvar Line Flows

If checked, this option will cause all Mvar flow text fields for lines to be displayed as the absolute value of the flow. Otherwise, the Mvar flow text fields will be positive near the source end of the line and negative near the sink end of the line.

### Use Absolute Values for MW Interface Flows

If checked, this option will cause all MW flow text fields for interfaces to be displayed as the absolute value of the flow. Otherwise, the MW flow text fields will be positive near the source end of the interface and negative near the sink end of the interface.

### Enable Mouse Wheel Zooming

When this box is checked, zooming can be done with a mouse wheel. The default is off.

**Show Online Hints**

When this box is checked, hovering the cursor over an object will briefly display information about the object in a popup hint.

**Visualizing Out-of-Service Elements**

Out-of-service elements can have optional visualization settings that makes them "stand out" in relation to in-service devices on the online diagram. The options for out-of-service devices includes **Use Dashed Lines**, **Draw and X Through Off-Line Generators**, and **Blink**. Additional controls for the Blink option are available, for setting the blink interval and color. Any combination of these options can be used for out-of-service elements, although the **X** option only applies to out-of-service generators.

**Browsing Path for Online Diagrams**

This option applies when you have [Online Links](#) included on a online diagram. Rather than specify the full path and name of a online diagram as a online link, you can specify the file name only. When the link is clicked in Run Mode, Simulator will check all directories listed here, in order, to try and find the online file name stored with the link.

---

## Pie Chart/Gauge Options

---

### Show Pie Charts/Gauges in Run Mode

When this option is checked, the pie charts and line gauges will be visible during Run Mode. Otherwise, they will be visible only during Edit Mode.

### Only Show Pie Charts/Gauges if relevant data exists

The relevancy of pie charts and gauges depends on the style of the pie chart or gauge. For MVA, MW, Mvar and Amp styles, the pie charts are considered irrelevant if the limits on the line or interface are 0. For max percent loading under contingency, the pie chart is irrelevant if no violation(s) under contingency occurred on the element. For the PTDF style, the pie chart is invalid if the PTDF value has not been calculated for the line or interface.

The options for pie charts and gauges are split into four sections, discussed in the additional topics:

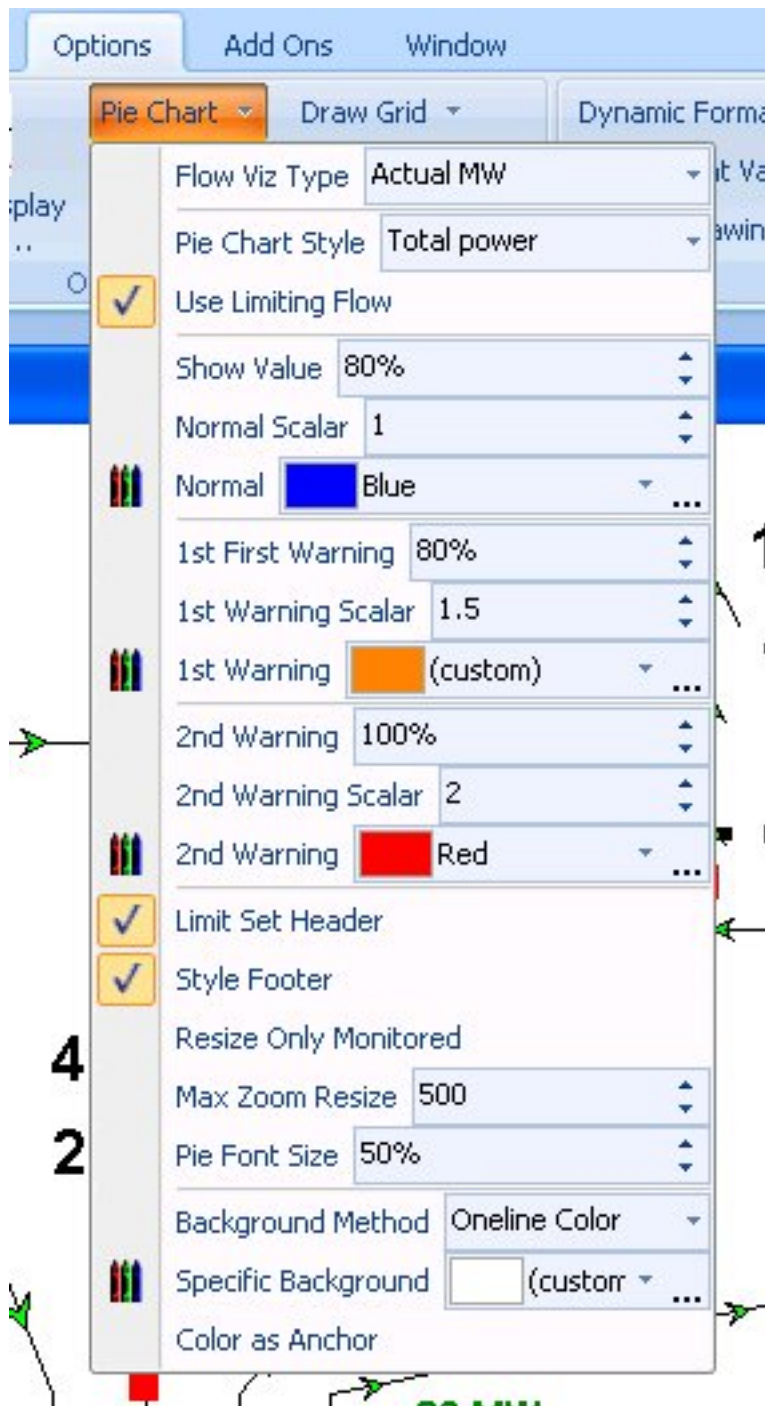
[Pie Charts/Gauges: Lines](#)

[Pie Charts/Gauges: Interfaces](#)

[Pie Charts/Gauges: Pie Chart/Gauge Styles](#)

[Pie Charts/Gauges: General Options](#)

Many of these same options are available on the Pie Chart menu found on the [Options ribbon tab](#).



---

## Animated Flows Options

---

The fields on the Animated Flows page of the [Online Display Options](#) dialog are used to customize the appearance of the animated flows on the oneline diagram.

### Show Flow Symbols

Determines whether power flows are animated on the onelines. If this option is not checked, then no flow symbols appear on the oneline.

### Show Flows On

Allows you to choose whether or not to display animated flows individually on Branch, Load, Generator and Shunt objects. Uncheck the box(es) for the objects that you do not wish to display animated flows.

### Base Flow Scaling on

Animated flows on onelines may depict either actual power flows or [power transfer distribution factors](#), depending on what you choose here. You can also choose to animate both the MW and MVAR flows or the MW and PTDF flows simultaneously. The Custom Float 1 option allows user specification of a value to depict in the flow animation. To use this option, values should be entered in the Custom Floating Point 1 field for the appropriate devices.

### Animate

This option determines whether or not the flow arrows are mobile or stationary on the oneline diagram. Unchecking this box will cause the flow arrows to remain stationary on the diagram.

### Scale Speed of Flow

Checking this box will cause the arrows to flow at a speed proportional to the represented quantity. When it is not checked, all of the flows will be at the same speed.

### Scale Size of Flow

If checked, the size of the animation symbols will vary to represent the magnitude of flow on the element (respective of the **Animation Parameter**.) Otherwise, the symbol size will be the same on all devices regardless of the magnitude represented.

### Reset Animated Flow Offsets

This button allows the user to reset the animated flows to start at a specified offset position. Mostly this would be used to reset the offset to 0, which would cause the animated flows to start at the beginning of the line or element. Since the animation moves the flow arrows every time the load flow is resolved, resetting flows to a specific offset could be useful for comparing different load flow solutions by looking at differences in the animated flow objects.

### Set Size, Density, and Reference Values for this oneline

Simulator detects the current zoom level and object parameters of the selected oneline diagram, and automatically adjusts the animation settings in an attempt to optimize the animation quality.

### Size

Determines the relative size of the animated flows on the devices. Increasing this number increases the size of the flow symbols. This field may range from 1 to 999.

**Density**

Determines the relative density of the animated flows on the devices. Increasing this value causes Simulator to display a greater number of flow symbols per unit distance on the oneline. This value may range from 1 to 999.

**Scaling Based on**

Determines whether the size and speed of animated flows represent actual flow or percentage loading. This option applies only to transmission lines and transformers as flows on other devices, such as loads and generators, always represent actual flow.

**Max. Zoom Level to Scale Size**

As a oneline is zoomed, the animated flow symbols increase in size. The value of Max. Zoom Level to Scale Size caps the size of the animated flows so that zooming beyond this level results in no further increase to the size of the flow symbols.

**Maintain Density above Maximum Zoom Level**

As a oneline is zoomed, the animated flow symbols decrease in density. Checking this box maintains the density of the animated flows after the maximum zoom level has been reached.

**Minimum Size in Pixels for In-service Elements**

Allows the specification of a minimum size for the animated flow objects for power systems elements that are in-service. The flow objects are sized based on the flow through the element. For elements with very small flows, it is often difficult to see the flow objects, and the minimum size can be set to make these flow objects more visible. The larger the minimum size value becomes, the more uniform the flow objects will become as elements are no longer sized based on their actual flow but rather the minimum size.

**Reference Values for Scaling**

Maximum flow reference for sizing the animated flows. The lower the MVA reference, the larger the animated flows appear on the oneline diagram as the actual flow value is compared to the reference value for scaling.

**Symbol Shape**

Simulator can display animated flows using directional arrows, circles, squares, or triangles.

**Animation Rate**

If the oneline animation is too fast or too slow, the animation rate can be adjusted by moving the slide bar.

**Symbol Fill Color**

Shows the fill color used for the animated flows if **Use Fill Color** is checked. Double-click on these fields to change the colors of the six different types of animated flows.

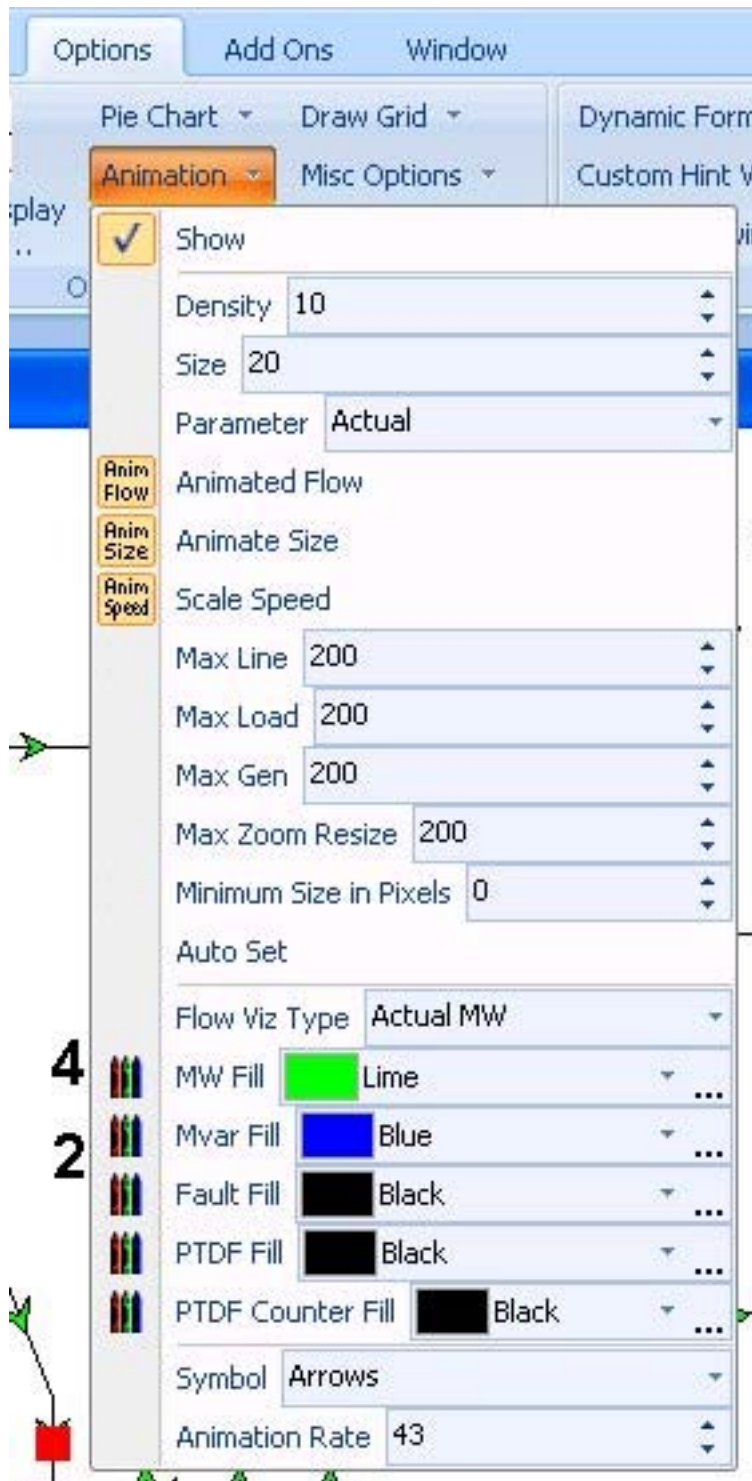
**Use Fill Color**

Checked if animated flow symbols should be filled using the **Fill Color**.

**Show PTDF Counter Flow**

Check this option if you wish for PTDF values that are in the opposite direction of the actual flow to be displayed using a different symbol color when visualizing PTDFs.

Many of these same options are available on the Animation Menu found on the [Options ribbon tab](#).





---

## Thumbnail View

---

The thumbnail view allows the user to see an overview of the online diagram in a smaller window in a specified location on the online diagram. The thumbnail view is useful when the user has zoomed in to a specific area of the online diagram, but still wants to see what part of the system they are observing on a larger scale as they pan around the diagram. The other application is to observe a more detailed part of the system in the thumbnail view as the user moves the cursor over the online diagram. The thumbnail view is not visible by default, but can be set up and displayed from the [Online Display Options](#) dialog. The options for the thumbnail view are:

### Show ThumbNail View

If checked, the thumbnail view will be visible in the specified corner of the online diagram.

### Size of ThumbNail View

The size of the thumbnail view as a percentage of the size of the online diagram.

### Zoom level of Thumbnail View

The amount to multiply the online diagram zoom level for display in the thumbnail view. The higher the multiplier, the more of the diagram you will see in the thumbnail view. You can instead indicate that the thumbnail view should display the diagram at a specified zoom level rather than a dynamically scaled zoom level based on the current diagram's zoom level.

### Location

Choose the location on the online diagram in which the thumbnail view is to appear.

### Border Width in Pixels

The pixel thickness of the border around the thumbnail view.

### Background Color

If **Use One-line Background Color** is checked, the thumbnail background color will be the same as the online diagram. If it is not checked, then a different thumbnail background color can be selected by clicking on the colored box next to the **Use Custom Background Color** label.

### Thumbnail view follows mouse cursor

This setting is very useful when is combined with the Zoom Level. Using a low multiplier and checking the Thumbnail view follows mouse cursor box will show a more detailed part of the system over which the cursor is being moved.

### Percent Transparent

This setting allows you to make the thumbnail window completely opaque, completely transparent, or some measure in between. The more transparent the thumbnail window, the more detail of the underlying online diagram is visible through the thumbnail window.

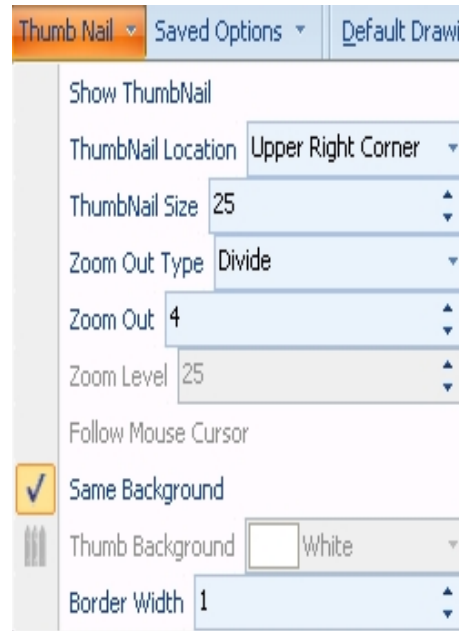
Many of these same options are available on the [Thumb Nail Options](#) menu found on the [Options](#) ribbon tab.

---

## Thumb Nail Menu

---

The Thumbnail View Options quick menu provides access to most of the options contained in the [Thumbnail View Options](#) dialog in an individual manner through menu commands. Select the **Thumb Nail** option from the **Online Options** ribbon group on the [Options](#) ribbon tab to access the menu.



This quick menu contains the following default buttons, in order from top to bottom:

- Show thumbnail on the screen
- Thumbnail location on the screen
- Size of Thumbnail View, as a percentage of the window size
- Zoom out type (divide or zoom level)
- Zoom out factor, defining the ratio of the diagram seen in the thumbnail relative to what is visible in the diagram itself.
- Enable/Disable Thumbnail View to follow mouse cursor
- Enable/Disable Using the Same Background as the main window
- Background Color, if not the same as the main window
- Line width of the border around the thumbnail window

---

## Display Object Options

---

### General Options (All) Tab

#### Percent of Injection Group/Owner object height used by name

This option allows the user to specify the height of the name with respect to the rectangle of the injection group graphical objects.

#### Display Circuit Breakers in Generators, Loads and Switched Shunts

Uncheck this box to hide circuit breakers from appearing in generators, loads, and switched shunts. If this option is checked, the user can use in **Display breakers on...** to select individually what elements can display circuit breakers.

#### Display Rotors in Generators

Uncheck this box to hide the rotors of the generators.

#### Change in Gen Rotor Angle per Refresh (degrees)

This value specifies the change in the rotor angle every time the animated flows refresh. This means that the greater the angle value, the faster the rotor will seem to rotate when the online is animated.

#### Show Field Inside Generator

Fields can be shown inside generator objects. Select a field from the drop-down box or use the Find button to search through the list of fields available for a generator. Set the field to *No Field* to not display any field inside generator objects. Set the **Digits** and **Decimals** for floating point fields. Adjust the **Relative Font Size** to change the size of the font relative to the generator object size. To ensure that the field fits entirely inside each generator object, check the option to **Dynamically decrease the font size so that the field fits inside the generator**.

#### Multi-Section Line Display Objects

##### Intermediate Bus Rotation Angle

Normally, the intermediate buses are perpendicular to the line. The value entered here can rotate the intermediate bus with respect to the original position. Therefore, a value of 0 degrees, will keep the intermediate buses perpendicular to the line, while a value of 90 degrees will set the intermediate buses aligned to the line.

##### Intermediate Bus Relative Size

This option is actually a multiplier for the size of the intermediate buses specified for each multi-section display object.

#### Bus Field Voltage Option

This option allows the user to specify in which units the bus fields with *Bus Voltage* as type of field will be displayed. This setting is only valid when the type of field selected is *Bus Voltage* in the [Bus Field Information](#) dialog. Choosing a voltage field in the Select Field will return the voltage value in the specific units of the field.

### Circuit Breakers Tab

#### Circuit Breaker Display Objects > Normally Closed

This section is used to specify the colors for all the circuit breakers normally closed in the oneline diagram. The colors that can be modified are the border and fill colors for presently closed and open circuit breakers. In order to change a color, click on the color box or on the **Change** button. Check **Use Fill** to use the specified fill color.

#### **Circuit Breaker Display Objects > Normally Open**

This section is used to specify the colors for all the circuit breakers normally open in the oneline diagram. The colors that can be modified are the border and fill colors for presently closed and open circuit breakers. In order to change a color, click on the color box or on the **Change** button. Check **Use Fill** to use the specified fill color.

#### **Shape**

Specify the shape of circuit breakers on transmission lines and transformers. The breakers shown on loads, generators, and switched shunts will always be the default rectangle shape.

---

## Substation Display Options

---

The **Substations** tab of the [Online Display Options](#) dialog provides parameters for customizing the appearance of [Substation](#) display objects. This tab is only available if the case contains at least one substation.

The upper right-hand corner of the Substation page displays a template of the substation object's appearance in terms of locations or "zones" where information can be displayed on the object. The rest of the page can be used to describe the appearance of the object and the information that appears in the substation objects.

### Upper % of Height

This percentage indicates how much of the object is populated by the "upper" zone of the object. This includes the *Upper Field*, *Upper Left (UL)* and *Upper Right (UR)*. As you decrease this percentage, the upper zone will get smaller, increasing the Identifier (middle) zone automatically, and vice-versa if you increase the percentage of the upper zone.

### Identifier % of Height

This is the percentage of the height of the substation object that is occupied by the identifier or "middle" zone. This is the substation identifier section of the object.

### Lower % of Height

This is the percentage of the substation object that is occupied by the "lower" zone of the object. This includes the *Lower Field*, *Lower Left (LL)* and *Lower Right (LR)*. Unlike the previous two percentages, this percentage cannot be directly modified. Instead, it is automatically determined based on the settings of the *Upper* and *Identifier* percentages.

### Left % of Width

This percentage indicates how much of the object is populated by the "left" zone of the object. This includes the *Upper Left (UL)* and *Lower Left (LL)*. As you decrease this percentage, the left zone will get smaller, increasing the Identifier (middle) zone automatically, and vice-versa if you increase the percentage of the left zone.

### Identifier % of Width

This is the percentage of the width of the substation object that is occupied by the identifier or "middle" zone. This is the substation identifier section of the object.

### Right % of Width

This percentage indicates how much of the object is populated by the "right" zone of the object. This includes the *Upper Right (UR)* and *Lower Right (LR)*. As you decrease this percentage, the right zone will get smaller, increasing the Identifier (middle) zone automatically, and vice-versa if you increase the percentage of the right zone.

### Buffer Percent

The buffer width for the height settings provide a buffer zone of the percentage width specified horizontally between the Identifier section and the upper / lower field sections. Similarly, the buffer width for the left and right width fields indicates a percentage buffer vertically between the upper / lower field zones and the four corner zones.

### Substation Identifier

Choose how the identifier will be displayed for the substation. Choose to display by name, by number, or by combinations of name and number. The identifier will be displayed in the *Identifier* or "middle" zone of the substation object.

## Shape

Select the shape used for substations that are not overriding the [Substation Layout Settings](#).

## What should be done when identifier text does not fit inside the width

As the name of this option suggest, you can choose how the text in the identifier section of the object should be modified if resizing of the substation object causes the text to be too large for the modified size.

## Extra Substation Fields

These four fields, two of which are represented as the *Upper Field* and *Lower Field* on the template, can be customized to display any substation field of your choosing. Use the drop-down arrow of the appropriate box to choose a field, or use the Find button to pull up a list of fields to choose from. Once the field has been chosen, set the digits and number of decimal places for the field.

## Upper-Left Symbol (UL)

Choose what symbol to display in the upper-left zone of the substation object. You can choose from None, Switched Shunt, Generator, Number of Buses, and Load. If the substation contains at least one type of the chosen object, a symbol representing that type of object will be displayed in the Upper-Left location of the object to indicate such a presence. By default, the Upper-Left zone will display the Generator symbol.

## Upper-Right Symbol (UR)

Choose what symbol to display in the upper-right zone of the substation object. You can choose from None, Switched Shunt, Generator, Number of Buses, and Load. If the substation contains at least one type of the chosen object, a symbol representing that type of object will be displayed in the Upper-Right location of the object to indicate such a presence. By default, the Upper-Right zone will display the Load symbol.

## Lower Left Symbol (LL)

Choose what symbol to display in the lower-left zone of the substation object. You can choose from None, Switched Shunt, Generator, Number of Buses, and Load. If the substation contains at least one type of the chosen object, a symbol representing that type of object will be displayed in the Lower-Left location of the object to indicate such a presence. By default, the Lower-Left zone will display the number of buses in the substation.

## Lower-Right Symbol (LR)

Choose what symbol to display in the lower-right zone of the substation object. You can choose from None, Switched Shunt, Generator, Number of Buses, and Load. If the substation contains at least one type of the chosen object, a symbol representing that type of object will be displayed in the Lower-Right location of the object to indicate such a presence. By default, the Lower-Right zone will display the Switched Shunt symbol.

---

## Online Animation

---

An important feature of PowerWorld Simulator is its support of animated onelines. The use of efficient display algorithms allow animation rates that are typically greater than several times per second, even on large cases and on onelines with a significant number of objects. The extensive use of animation makes the display "come alive" so that system conditions can be ascertained more easily.

In Simulator, animation is started from Run Mode by selecting the **Play** button in the [Power Flow Tools](#) ribbon group on the [Tools](#) ribbon tab. In Viewer, animation is started automatically when you load a case.

The animation can be controlled and customized from the [Animated Flows Tab](#) of the [Online Display Options](#) dialog. To access this dialog, select **Online Display Options** from the **Online Options** ribbon group on the [Options](#) ribbon tab, or right-click on an empty area of the online diagram and select **Online Display Options** from the resulting [local menu](#).

---

## Copying Onelines to Other Programs

---

The onelines can be easily copied to other programs. This allows you to add PowerWorld onelines to your word processor documents or slide presentations. The simplest way to copy a oneline diagram to another program is to use the Windows Clipboard. This is accomplished as follows:

- In Simulator or Viewer, zoom and/or pan the display to the portion of the oneline diagram you would like to copy.
- Right-click on an empty portion of the oneline to display the [local menu](#).
- Select **Copy Image to Clipboard** menu item. This places a copy of the oneline into the Window's clipboard.
- In the other program, use **Paste** or **Paste Special** to copy the contents of the clipboard into that program. The oneline is pasted into the program using the Metafile format.

In addition, oneline diagrams can also be [saved as image files](#) by right-clicking on an empty portion of the diagram and choosing **Export Image to File**.



---

## Display Explorer

---

The Display Objects case information display presents data describing each graphical object in the online diagram. This display functions in the exact same manner as the [Model Explorer](#) and is thus called the **Display Explorer** instead. The Display Explorer contains [Case Information Displays](#) and therefore can be used in a manner consistent with all other case information displays. It has a [case information toolbar](#) from which you can print, copy, and modify its records as well as view the information dialog of its associated power system object if it has any. You can also sort the object records by clicking on the heading of the field by which you want to sort. Additionally, when Simulator is in [Edit Mode](#), the toolbar **Records** menu allows you to remove existing objects from the online diagram.

To show the Display Objects case information display, go to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group select **List Display > All Display Objects**.

As mentioned above, the Display Explorer functions identically to the [Model Explorer](#). The only differences in functionality are due to additional options found on the bottom left of the dialog. This offers the following additional features.

### Show Only Objects Selected

By checking this box, only the selected display objects currently selected in the online diagram will be shown.

### How to List Grouped Objects

This specifies how to list grouped objects if there are any in the online diagram. The choices are to list individually the components of the group, to list only the groups, or to list both the individual elements and the groups. If there are not grouped objects in the online diagram, this control is disabled.

### Save Complete Display to AXD

By clicking this button, all the objects that are shown in the case information display will be saved to a [display auxiliary file](#).

By default, the display object records case information display contains the following fields:

#### Type

The type of display object.

#### X/Longitude Location, Y/Latitude Location

The x and y coordinates of the object's location.

#### Layer Name

The layer name to which the display object is assigned.

#### Layer Shown, Selectable in Edit Mode, Low Zoom Level, High Zoom Level

These fields only display information about the layer to which the display object belongs. Therefore, the values for these fields can only be modified through the [Screen Layer case information dialog](#) or the [Screen Layer dialog](#).

#### Anchored

This field indicates whether or not the display object is an [anchored object](#). If the display object can not be an anchored object, then the field is empty.

#### Font Size

For text objects, this field shows the font size.

---

## Supplemental Data

---

Supplemental Data can be used to identify display objects that do not represent power system elements. User defined data types can be created and linked to display objects for use in filtering, dynamic formatting, Select by Criteria, and custom hints.

### Supplemental Classification Definition

A Supplemental Classification is used to group Supplemental Data records into categories. As an example, if using Supplemental Data records to identify geographical borders, a Supplemental Classification could be *Country*. Each Supplemental Data record uses exactly one Supplemental Classification record.

Supplemental Classification records are defined by one key field:

#### Name

This field uniquely identifies the classification and cannot be blank.

### Supplemental Data Definition

Supplemental data records are defined by two required key fields:

#### Classification

This is the Supplemental Classification Name. This field cannot be blank.

#### Name

This is a unique identifier used to further define the Classification. If using supplemental data records to identify geographical borders, the Name could be *United States* to go with a Classification of *Country*. This field cannot be blank.

In addition to the Classification and Name, geographic information, expressions, and custom strings, integers, and floating point values can be assigned to supplemental data records. This information can be entered in the Supplemental Data Case Information Display discussed below.

### Displaying Supplemental Data Records

Supplemental Data records are displayed in the Supplemental Data Case Information Display found under **Case Information > Other Data > Supplemental Data**. This display consists of two tabs, **Supplemental Classification** and **Supplemental Data**. The table on the Supplemental Classification tab lists all of the Classifications that have been defined. The table on the Supplemental Data tab lists all of the Supplemental Data records that have been defined and provides the means for creating new Supplemental Data records AND Supplemental Classification records. Supplemental Classification records cannot be created under the Supplemental Classification tab.

### Creating Supplemental Data Records

To create a new Supplemental Data record, from the **Supplemental Data** tab, right-click on the table and choose **Insert** from the local menu. The **Define Supplemental Data** dialog will open.



Either select an existing **Classification** from the drop-down box or create a new Classification by clicking **New**. (If creating a new Classification, another dialog box will open with a prompt for the Classification Name.) Type in a **Name** for the record. To create the record and close the dialog click **OK**. To create the record without closing the dialog click **Save**. To exit the dialog without creating the record click **Cancel**.

### Linking Display Objects to Supplemental Data

Display objects that cannot be linked to a power system element can be linked to a Supplemental Data record. These display objects include Background Line, Background Rectangle, Background Ellipse, Background Picture, Text, Case Information Memo, Online Field, Online Link, Document Link, Memo Text, and Generic Model Field.

Linking any of these objects to supplemental data must be done through the **Display Explorer** found under **Onelines > All Display Objects**. From the **Explore** tab, choose the display object type to link. If they are not already displayed, the **Supplemental Data Classification** and **Supplemental Data Name** fields must be added to the list of fields in the table. Simply enter the appropriate Classification and Name in the fields to link a particular display object to a Supplemental Data record.

Once a display object is linked to a supplemental data record, the information in the supplemental data record can be used to filter, dynamically format, Select by Criteria, and display custom hints for that display object.

Because there is no requirement to link display objects that cannot be linked to power system elements to supplemental data records, these objects will not be highlighted along with other unlinked elements when they are not linked to a supplemental data record.

### Automatic Linking of Supplemental Data When Inserting Geographic Information

Background objects (lines and ellipses) created from inserting objects from borders included with Simulator and shapefiles can be linked automatically to supplemental data records.

When [auto inserting borders](#) included with Simulator, the option to **Automatically link county and state borders to supplemental data** needs to be checked for this to occur.

When inserting background objects from shapefiles, there are several options that need to be set for proper linking to supplemental data. These options are available on the **Identify** tab of the [GIS Shapefile Data Dialog](#).

---

## Supplemental Data Fields on Onelines

---

Supplemental Data Field objects represent information about [Supplemental Data](#) records. The information that can be represented in these fields is the same in either Edit or Run Mode.

Supplemental Data Fields can be added to a oneline while in Edit Mode from **Draw > Field > Supplemental Data Field**. After selecting this option, move the cursor to the location on the oneline where the new field should be placed and left-click. The Supplemental Data Field Options dialog will open.

The **Supplemental Data Field Options** dialog can be used to modify the properties of individual supplemental data fields on the diagram. This dialog contains the following fields:

### Classification

Select the Supplemental Classification from the drop-down list of available classifications.

### Name

Select the Name of the Supplemental Data record from the drop-down list of available names. This list is populated with the appropriate names based on the selected Classification.

### Total Digits in Field

Total number of digits to show in the field.

### Field Prefix

A prefix that can be specified and displayed with the selected value.

### Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

### Field Value

The current value of the field being displayed.

### Rotation Angle in Degrees

The angle at which the text will be displayed on the diagram.

### Type of Field

Select a field from the drop-down list of available fields or click **Find Field** to search through the list of available fields.

Select **OK** to close the dialog and save the changes, or click **Cancel** to close the dialog without saving the changes.

---

## **Movie Maker**

---

Simulator version 13 no longer contains a built-in movie maker tool. Instead, PowerWorld Corporation recommends other movie making tools that can be used from outside of Simulator. Please refer to the PowerWorld website at [www.powerworld.com/support/videocapture.asp](http://www.powerworld.com/support/videocapture.asp) for references to these third party movie tools.

---

## Map Projections

---

### Background on Map Projections

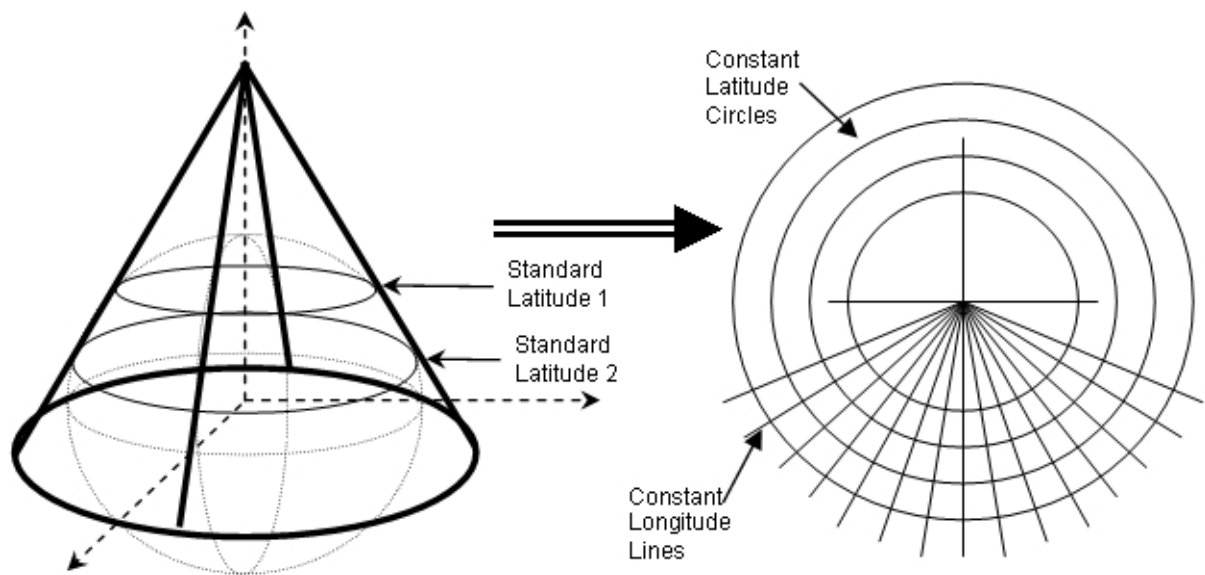
In order to create a map of a spherical object such as the Earth it is first necessary to convert the Earth coordinates of latitude and longitude into a two-dimensional coordinate system needed for a map. This conversion from the spherical coordinates of latitude and longitude to some xy Cartesian coordinate system is called a *map projection*. Mapmakers and mathematicians throughout history have invented many map projections. For each map, the mapmaker must decide which projection will look best for his or her purposes. This will depend on what part of the globe is being drawn, the size of the area being drawn, and the shape of the object being drawn. In general, map projections fall into three standard classes: cylindrical, conical, and planar. For a complete analysis of the topic of map projections see reference 1.

For maps of North America, PowerWorld supports a Simple Conic map projection which is described below.

For maps of the entire Earth, PowerWorld supports a Mercator projection which is described below.

### Simple Equidistant Conic Map Projection

For maps made of the 48 conterminous states in the United States, maps are often made using one of the conical map projections. These projections can be recognized by the constant latitude lines that are seen as arcs. When looking at the U.S./Canadian border in the western United States, you will see that it curves on maps made using a conical projection, even though this is a constant latitude border. Several conical projections exist including the Albers Equal-Area Projection, Lambert Conformal Projection, and Simple Equidistant Projection. They can be visualized as taking the globe and placing a cone over it as shown in Figure 1 and then using this to map the coordinates to the xy plane shown at the right. Constant longitude lines are seen as straight lines and constant latitude lines are seen as arcs on the resulting map.



**Figure 1 Conic Map Projection**

The equations for converting back and forth between (longitude,latitude) and (x,y) using the simple equidistant projection are described next. These equations are taken from reference 1.

First, you must specify seven values that are parameters of the mapping:

- $R$  = Radius of the sphere (earth) in the units used on the map
- $\phi_o$  = The latitude of the "shifted" origin
- $\lambda_o$  = The longitude of the "shifted" origin
- $\phi_1$  = The first standard parallel
- $\phi_2$  = The second standard parallel
- $x_f$  = The shift in the x-direction. Referred to as "false-easting" in some map projections. This is done so that all coordinates are positive.
- $y_f$  = The shift in the y-direction. Referred to as "false-northing" in some map projections. This is done so that all coordinates are positive.

For these values, the following variables are defined:

$$\rho_o = R(G - \phi_o) \qquad G = \frac{\cos \phi_1}{n} + \phi_1 \qquad n = \frac{\cos \phi_1 - \cos \phi_2}{\phi_2 - \phi_1}$$

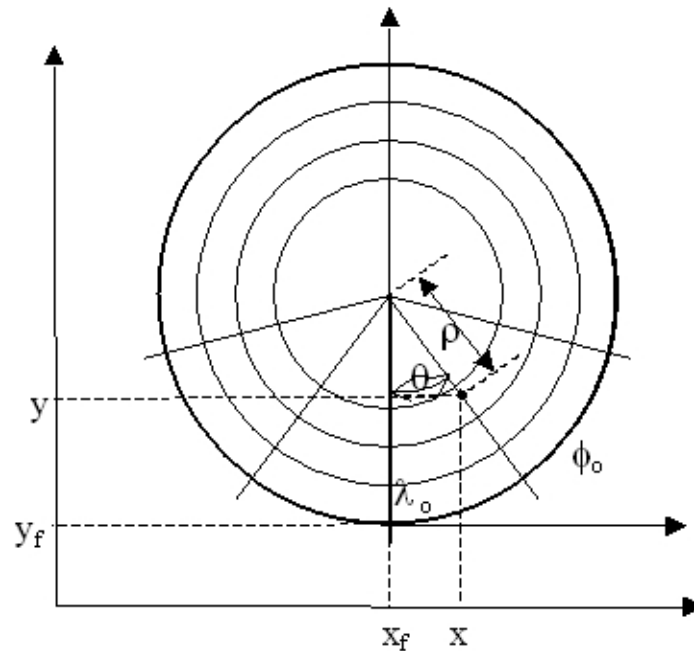
Then given a latitude =  $\phi$  and a longitude =  $\lambda$ , calculate the x and y coordinate using

$$\begin{aligned} \rho &= R(G - \phi) \\ \theta &= n(\lambda - \lambda_o) \\ x &= \rho \sin \theta + x_f \\ y &= \rho_o - \rho \cos \theta + y_f \end{aligned}$$

And given an x and y coordinate, calculate the latitude( $\phi$ ) and longitude( $\lambda$ ) using

$$\begin{aligned} \rho &= \pm \sqrt{(x - x_f)^2 + (\rho_o - y + y_f)^2} \\ &\text{taking the sign of } n \\ \theta &= \arctan \left[ \frac{x - x_f}{\rho_o - y + y_f} \right] \\ \phi &= G - \frac{\rho}{R} \\ \lambda &= \lambda_o + \frac{\theta}{n} \end{aligned}$$

Figure 2 shows more clearly what some of these variables are.



**Figure 2 Defining Variables for Conic Projections**

Maps made using PowerWorld's built-in map borders for North America use the following parameters:

$$\begin{aligned}
 R &= 30700.0 \\
 \phi_0 &= +9.7 \text{ degrees} \\
 \lambda_0 &= -96.5 \text{ degrees (negative because in Western Hemisphere)} \\
 \phi_1 &= 0.00 \text{ degrees} \\
 \phi_2 &= 50.00 \text{ degrees} \\
 X_f &= 20968 \\
 Y_f &= 0
 \end{aligned}$$

These numbers were thus set to be the default map projection numbers within PowerWorld. This projection will work well for the one-line diagrams with PowerWorld Corporation has created on the same standard background.

### **Mercator Projection Use in PowerWorld**

For the Mercator Projection, PowerWorld assumes  $R = 30700.0$  units. From this, given a latitude  $= \phi$  and a longitude  $= \lambda$ , calculate the  $x$  and  $y$  coordinate using

$$\begin{aligned}
 x &= R * \lambda \\
 y &= R * \ln \left[ \tan \left( \frac{\pi}{4} + \frac{\phi}{2} \right) \right]
 \end{aligned}$$

And given an  $x$  and  $y$  coordinate, we calculate the latitude( $\phi$ ) and longitude( $\lambda$ ) using



$$\lambda = \frac{x}{R}$$

$$\phi = \tan^{-1} \left[ \sinh \left( \frac{y}{R} \right) \right]$$

## References

1. Snyder, John P., Map Projections - A Working Manual, U. Geological Survey Professional Paper 1395, United States Government Printing Office, Washington, D.C.: 1987.

---

## Great Circle Distance Dialog

---

To find the Great Circle Distance between any two points of longitude, latitude, select **GIS Tools > Great Circle Distance** from the [Active](#) ribbon group on the [Onelines](#) ribbon tab and the Great Circle Distance Dialog will be displayed. This can be done in Edit or Run Mode.

The Great Circle Distance Dialog allows the calculation of distance between two points of longitude, latitude. The [Great Circle Distance Calculation](#) assumes that the earth is a sphere and makes no adjustments for the actual flattening of the earth. The two longitude, latitude points can either be entered manually for any two valid points or can be chosen from valid longitude, latitude values stored with either buses or substations. Valid longitude values are between -180 and 180 degrees. Valid latitude values are between -90 and 90 degrees. The calculated distance is given in either kilometers or miles.

### From Location

Click the **Choose From Location** button to choose either the location of a bus or substation as the **From Location**. The **Latitude (degrees)** and **Longitude (degrees)** entries for the From Location will be filled in with the latitude, longitude information that is stored with the chosen element in the case data. These entries are not based on the location of a display object, if one exists, of the chosen element. If the element does not have valid geographic information stored, the values that are entered for the latitude, longitude will not be valid.

The Latitude, Longitude entries can also be entered manually.

### To Location

Click the **Choose To Location** button to choose either the location of a bus or substation as the **To Location**. The **Latitude (degrees)** and **Longitude (degrees)** entries for the To Location will be filled in with the latitude, longitude information that is stored with the chosen element in the case data. These entries are not based on the location of a display object, if one exists, of the chosen element. If the element does not have valid geographic information stored, the values that are entered for the latitude, longitude will not be valid.

The Latitude, Longitude entries can also be entered manually.

### Units

Choose the units in which to report the calculated distance.

### Calculate Distance

Click this button to calculate the distance in the chosen Units between the From Location and the To Location. The calculated distance will appear in the box above the button. *Error in input data* will be returned if there is a problem with any of the latitude, longitude information.

---

## Great Circle Distance Calculation

---

The Great Circle Distance Calculation allows the calculation of distance between two points of longitude, latitude. The Great Circle Distance Calculation assumes that the earth is a sphere and makes no adjustments for the actual flattening of the earth.

The calculation consists of the following parameters:

- $R$  = Radius of the Earth (6378.137 km from NAD83/WGS84 datum)
- $\phi_0$  = The latitude (radians) of the starting location
- $\lambda_0$  = The longitude (radians) of the starting location
- $\phi_1$  = The latitude (radians) of the ending location
- $\lambda_1$  = The longitude (radians) of the ending location
- $d$  = Distance in kilometers between starting and ending location

The distance in kilometers between the starting and ending location is determined from the following equation:

$$d = 2R \arcsin \left( \sqrt{\sin^2 \left( \frac{\phi_0 - \phi_1}{2} \right) + \cos(\phi_0) \cos(\phi_1) \sin^2 \left( \frac{\lambda_0 - \lambda_1}{2} \right)} \right)$$

This calculation is used with several GIS tools to determine the distance between two locations or facilities on a diagram.

---

## Populate Lon,Lat with Display X,Y

---

To populate the longitude,latitude fields of buses and substations with the locations of corresponding display objects converted to longitude,latitude using the selected projection, go to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **GIS Tools > Populate Lon,Lat with Display X,Y**.

The Populate Lon,Lat with Display X,Y Dialog will be displayed. This can be done in either Edit or Run Mode. The dialog will show which map projection is currently in use for the oneline. Users should be careful to not change the map projection once one has been established so that all objects on the oneline are consistently using the same projection. If no projection has been set, the dialog shows **None (x,y)** as the Map Projection, selecting a map projection will establish that map projection as the current oneline map projection.

### [Map Projection](#)

#### **None (x,y)**

No map projection is in use.

#### **North American (simple conic)**

This projection is better when dealing with longitude, latitude points in North America. This is the projection that has been used in Simulator for years in defining geographic borders and placing them on a oneline.

#### **Entire World (Mercator)**

This projection is better when dealing with longitude, latitude points spread throughout the world.

Click **Populate Longitude,Latitude** to complete the conversion. The longitude, latitude values converted from display x, y coordinates can be viewed on the bus and substation case information displays or the information dialogs.

---

## Geography/Coordinates

---

The option is available to display screen coordinates in longitude, latitude instead of x,y. This option can be set by selecting **Online Display Options** from the [Onelines](#) ribbon tab or the [Options](#) ribbon tab. The following options are located under the **Geography/Coordinates** category.

### [Map Projection](#) in Use

The map projection indicates which projection has been selected for inserting objects on the online. Users should be careful to not change a projection once one has been established. Otherwise, there is the risk that different objects will be inserted with different projections.

### **Show longitude, latitude coordinates when showing x, y coordinates**

When this option is checked and a map projection other than *None (x,y)* is selected, longitude, latitude coordinates will be displayed anywhere that x,y screen coordinates are normally displayed. This option will only be enabled when a map projection other than *None (x,y)* has been selected.

---

## Shape File Import

---

Simulator allows you to import ESRI shapefiles (\*.shp/\*.dbf pairs) as a group of background lines or points on a oneline diagram. There are several locations where you may open the GIS Shapefile Import Dialog.

- Go to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **GIS Tools > Insert GIS Data from Shapefile**.
- Go to the [Draw](#) ribbon tab, then on the [Individual Insert](#) ribbon group choose **Background > Insert GIS Data from Shapefile**
- Go to the [Draw](#) ribbon tab, then on the [Quick Insert](#) ribbon group choose **Auto Insert > Insert GIS Data from Shapefile**

The GIS Shapefile Data Dialog has the following sections:

[Control](#)

[Modify](#)

[Identify](#)

[Format](#)

[Shape List](#)

---

## GIS Shapefile Data: Control

---

This is the first page of the [GIS Shapefile Data](#) dialog. The control page contains the controls for reading a shapefile into Simulator and provides some basic information on the shapes read from the file.

### Read in Shapefile

Pressing this button will allow you to choose a shapefile and will load it into Simulator for placement on the currently selected online diagram.

### Save Shapefile

Clicking this button will save to file the shapefile information currently loaded in memory. This option is useful if making changes to shape format attributes or eliminating certain shapes from those read in and then saving these changes in the shapefile format.

### Shapefile Information

Once the shapefile has been loaded into memory, this section will be populated with general information on the number of shapes loaded, and the maximum and minimum X and Y values (usually in longitude and latitude.)

### Transfer Shapefile Objects to Online

If you are ready to place the shapes on the online diagram, but want to continue working with existing shapes or read more shapes from another file, press the **Transfer Shapes to Online and Clear** button. This will keep the dialog open following the transfer. It also gives you options of what should happen to the current shapes in memory after you have transferred them to the diagram. You can choose to clear only the shapes that were transferred, all of the shapes, or none of the shapes. The first choice relates to options you have on the [Shapefile Objects](#) page regarding whether or not to transfer certain shapes from the shapefile to the diagram.

### Insert Into Layer

Before transferring the shapes to the online diagram, you can choose which [layer](#) the shapes should be assigned to. If you wish to place the shapes into a new undefined layer, click on the **Define Layer** button to access the [Screen Layers](#) table.

### Transfer Shapes to Online and Close

If you wish to transfer the currently loaded shapes to the online and close the dialog (clearing the shapes from memory), press this button to complete the process.

---

## GIS Shapefile Data: Identify

---

After using the [Control](#) page to read in shapes from a shapefile but before placing them on the diagram, identification options can be specified on the Identify page.

### Identification String for Display Auxiliary Files

Shapefile shapes are created as background objects when added to a diagram. If using [display auxiliary files](#) to create or edit diagrams, objects must be uniquely identified. Because background objects typically do not link to an object in the power flow case, and therefore have no other unique means of identification, a unique ID string must be assigned to objects created via shapefiles. This unique identifier populates the **Auxiliary ID** field used in [display auxiliary files](#) and can be viewed from the [Display Explorer](#).

#### Prefix

Optionally, a prefix can be concatenated with the selected Attribute to further identify the shape.

#### Attribute

The drop-down box is populated with the fields that are available with the .dbf file. Choose a field whose value will be used to uniquely identify the shape. By default, the *Record Number* of the shape is used for identification if no other Attribute is selected.

#### Suffix

Optionally, a suffix can be concatenated with the selected Attribute to further identify the shape.

### Linkage to Supplemental Data

Background objects created from shapefiles can be linked to [Supplemental Data](#) records for purposes of further identifying the objects. This is useful when applying filters, applying dynamic formatting, using Select By Criteria, or defining custom hints with display objects created from shapefiles.

#### Link Display Objects to Supplemental Data

Check this box to link the background objects created from shapefile shapes to Supplemental Data records.

#### Supplemental Classification

Select an existing Supplemental Classification from the drop-down box or click **New** to create a new Supplemental Classification.

#### Attribute for Supplemental Data Name

Select an Attribute from the drop-down box. The list of attributes is populated from the fields that are available in the .dbf file. For each unique Attribute value, a supplemental data record will be created with the chosen Supplemental Classification and with its Name set to the Attribute value.



---

## GIS Shapefile Data: Modify

---

After using the [Control](#) page to read in shapes from a shapefile but before placing them on the diagram, the Modify page can be used to modify the XY attributes of the shapes.

### [Map Projection](#)

There are two map projections you can choose from when placing the shapes onto a online diagram. The choices are:

- North American (simple conic)
- Entire World (Mercator)

Once you have selected which projection to use, click on the **Convert to Specified Map Projection** button to process the conversion on the shapes currently in memory.

### **Shift/Scale Shapefile X/Y Data**

You also have the option to shift the XY coordinates of shapes or scale them by scalar values. To shift shapes, enter scalar shift values for the X value and Y value and press the **Shift XY Data** button. To scale the XY coordinates, enter scalar scaling factors for the X and Y value, and press the **Scale XY Data** button.

---

## GIS Shapefile Data: Format

---

After using the [Control](#) page to read in shapes from a shapefile but before placing them on the diagram, the Format page can be used to modify the appearance attributes of the shapes.

### Change Shape Format Characteristics

#### Point Size

The size of points read from the shapefile can be adjusted when transferred to the oneline diagram.

#### Line/Border Thickness

Choose the pixel thickness of the shapefile lines when transferred to the oneline diagram.

#### Line/Border Color

Set the line color of the shapefile lines when transferred to the oneline diagram.

#### Fill Color for Points/Polygons

Set the fill color of the shapefile points and polygons when transferred to the oneline diagram. The box labeled **Use Fill Color** must be checked as well.

#### Stack Level

Indicate which [stack level](#) should be applied to the shapes when they are transferred to the oneline diagram.

#### Immobile

Check this box if you wish for the shape file objects to be immobile once they are transferred to the oneline diagram. This will prevent you from inadvertently selecting and moving the shapefile objects on the oneline diagram while in [Edit mode](#).

#### Set all shape format attributes to the values above

Once you have finished changing the format settings for the shapes, press this button to apply them for the transfer of shapes to the oneline diagram.

### Automatic Color Mapping

#### Available Attributes

To customize the shapefile shapes based on a field value read from the .dbf file, select an attribute from the list.

#### Color Map

Select a color map to apply to the selected attribute. To modify existing or create new color maps, click on the **Modify Color Maps** button. The color of the object will then be based on where its selected attribute value maps to the selected color map.

#### Change Border Color, Change Fill Color

These two boxes allow you to choose if the color map selected should apply to the border color, the fill color, or both.

#### Change Color Based on Attribute/Color Map

Once you have finished setting the color map for a selected attribute, press this button to apply the settings before the transfer of shapes to the oneline diagram.

---

**GIS Shapefile Data: Shape List**

---

After using the [Control](#) page to read in shapes from a shapefile but before placing them on the diagram, the Shape List page can be used to flag shapes for inclusion in the transfer and customize the appearance attributes of specific shapes.

This page will be blank until you have actually read a shapefile into memory. Once you have done so, you will see a list of all shapes read from the file. The table allows you to customize the appearance and attributes of a specific shape, similar to the options on the [Format](#) page.

In addition, you also can choose to include or exclude shapes from the transfer to the online diagram. This gives you the greatest flexibility for hand-picking which shapes you want to add to a online diagram.

---

## Shapefile Database Record Dialog

---

The Shapefile Database Record dialog is available by selecting **Show Shapefile Fields...** from the local popup menu (obtained by right-clicking on a display object) of any display object that was created from a shapefile. This menu option will not be available for display objects that were not created from a shapefile. This dialog provides information read from the database file (.dbf) associated with the shapefile from which the display object was created. Each time that this dialog is displayed, the database file is read to fill in the field information. The field information is not stored in the PowerWorld display file (.pwd) so the associated database file must be available for this information to be displayed.

### File Name

This is the path and name of the database file (.dbf) associated with the shapefile from which the display object was created.

### Record Number

This is the record number of the display object within the shapefile.

### List of Fields

This list details the fields and values belonging to the given record number in the database file.

### Edit Oneline Browsing Path

Select this option to update the Oneline Browsing Path. This browsing path will be used to search all listed directories for the database file specified in File Name if it cannot be found in the directory specified with the File Name.

---

## Export Online As Shapefile

---

Display objects can be exported into shapefile formats (.shp/.dbf pairs) by going to the [Onelines](#) ribbon tab, then in the [Active](#) ribbon group, choosing **GIS Tools > Export Online as Shapefile**. This will open the Select Objects and Fields to Export Dialog, which is similar to the Select by Criteria Dialog.

The dialog is used to select one type of display object to export to the shapefile and the criteria used to select which specific objects will be included in the export.

### Selecting Areas and Zones

Use the **Area** and **Zone** fields to select the areas and zones in which to select display objects. [Ranges](#) of area and zone numbers can be entered in the usual way, or the **All Areas** or **All Zone** boxes can be checked to select all areas and all zones respectively. The **Areas** and **Zones** tab pages can also be used to select the areas and zones in which to select display objects.

### Selecting Voltage Levels

Specify the max and min voltage levels using the boxes that are provided, keeping in mind that all voltages are in kV. The **All Voltages** box can be checked to include all voltages without requiring the specification of a voltage range.

### Selecting Layers

All layers can be selected by choosing the **All** option for Layers. If specific layers are to be selected, the **Range** option should be selected and the **Layers** tab should be used to check and uncheck specific layers in which to select display objects.

### Type of Drawing Object

Select the type of object to export in the **Type of Drawing Object** dropdown box. Alternatively, the **Find...** button next to the dropdown can be used to select the object type from a list of available object types.

### Filter

The **Filter** dropdown box provides a list of filters available for the selected Type of Drawing Object. The **Find...** button next to this dropdown can be used to find a filter or define a new filter for the selected object type. Filter options will not be enabled when an object type that does not allow filtering is selected.

### Selecting Fields

The **Fields** list will be populated for object types that have associated fields. The selected fields will be written to the database file (.dbf) along with the value of the field. When the **Specified** option is selected, individual fields can be chosen by checking the box next to specific fields. When the **All** option is selected, then all fields associated with the selected type of object will be included in the database file. When the **Show Only Commonly Used Fields** option is checked, only those fields that are deemed to be commonly used will be displayed in the list. To show all fields associated with the selected object type, uncheck this box. Use the **Check All** and **Uncheck All** buttons to check all or uncheck all of the fields.

### Advanced Selection Criteria

If only objects that are currently visible on the display (due to layering, etc.) are to be selected, check the box labeled **Select only currently visible objects**. If some display objects have been selected prior to opening the dialog, an advanced selection option

becomes available. If the box labeled **Use as a filter on presently selected objects** is checked, the criteria chosen in the dialog will only affect the previously selected objects. In other words, only the objects that were previously selected AND that match the chosen criteria will remain selected when exporting to the shapefile.

### **Export Coordinates in Longitude, Latitude**

If a valid map projection has been selected for displaying display object coordinates in longitude, latitude, then the option to **Export coordinates in longitude, latitude** will be enabled. (A valid map projection can be set with the Map Projection in Use Option found in the [Geography/Coordinates](#) category of the Online Display Options dialog.) By default, the Export coordinates in longitude, latitude option will be checked when it is enabled. When checked, the coordinates of the display objects will be exported to the shapefile in longitude, latitude instead of PowerWorld Simulator x,y coordinates.

### **Saving Shapefile Export Descriptions**

Shapefile Export Description settings can be saved with the case by clicking **Save As** to save with a new name or **Save** to save with the current name. This will allow the quick recovery of settings that may have been previously used. Use the **Rename** button to rename an already saved set or the **Delete** button to remove a previously saved set of criteria. If Shapefile Export Description sets have been saved, they can be exported to an auxiliary file by choosing **Save to AUX file**. The **Load from AUX file** button will load an entire auxiliary file regardless of whether it contains Shapefile Export Descriptions or not.

### **Saving the Shapefile**

After all criteria have been set, click **OK**. A dialog will open prompting for the name of the file to save. Enter a file name and click **Save**. The type of file that is specified in the dialog is a shapefile with .shp extension, however, three files will be saved. All files will have the name specified in the save dialog, but the extensions will be .shp, .dbf, and .shx.

---

## Insert Measure Line

---

Distances between display objects can be approximated by inserting a measure line on the display. A measure line is similar to a background line. It can be drawn as a straight line between two points or as a line with many vertices and segments. By default, a measure line is drawn as a yellow line with a pixel thickness of 4.

To insert a measure line go to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **GIS Tools > Insert Measure Line**. Then position the mouse cursor on the display where the line should begin and click and release the left mouse button. Move the mouse to the desired termination point of the first line segment. A straight segment will follow the mouse movements. Click and release the left mouse button to complete the line segment and to prepare for drawing the next segment, or double-click if this is the last line segment. To draw a freehand shape rather than a series of straight line segments, click and hold the left mouse button where the freehand shape begins and drag the mouse to trace the desired shape (while holding the left mouse button down). Release the left mouse button to complete the section of the freehand shape. At this point, either another freehand section or a straight line segment can be added. When all desired freehand and straight line segments have been added, double-click the left mouse button to complete the line.

To change the shape of the line, first left click on the line to select it. Handles will appear at each vertex. A vertex can be moved by holding the left mouse button down and dragging the vertex to a new location. To remove a vertex, hold down the CTRL key and then click the vertex you would like to delete. To add a vertex, hold down the CTRL key and then click on the line where the new vertex should be added. Note that freehand lines are nothing more than a continuous series of vertices.

Once the measure line has been added to the display, the length of the line can be displayed by right-clicking on the line and selecting **Measure Length...** from the local menu. The units of the length are dependent on the map projection in use and whether the option to **Show longitude,latitude coordinates when showing x,y coordinates** is checked on the [Online Display Options dialog](#). If the map projection in use is something other than  $x,y$  and the option to show longitude,latitude coordinates is checked, the length will be given in miles and kilometers. Otherwise, the length will be given in Simulator units. The [Great Circle Distance calculation](#) is used to determine the distance between vertices on the line to determine the approximate length in miles and kilometers.

---

### Delete All Measure Lines

---

All [measure lines](#) can be deleted from the display. To do this go to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **GIS Tools > Delete All Measure Lines**.



---

## Path Distances from Bus or Group

---

To access the Determine Path Distances from Bus or Group dialog go to the [Tools](#) ribbon tab, and choose **Connections > Determine Path Distances to Bus** from the [Other Tools](#) ribbon group. The Determine Path Distances from Bus or Group dialog provides a way to calculate how far electrically every bus in the system is from a particular starting point.

After clicking the **Calculate** button, all buses that are part of the start element will be labeled internally as being a path distance zero. Then a calculation will be done that determines for *each bus* in the system the shortest total path length in getting from each bus to the Start Element. Buses that cannot reach the Start Element will be flagged with a very large distance (by default 10,000). The results of the calculation will then be copied into the Bus Field specified under the Bus Field to Populate option. The results will be displayed in a case information display at the bottom of the dialog.

The options on the dialog are described below.

### Start Element Type and Element

Choose an element type to be either a Bus, Substation, Area, Zone, Super Area or Injection Group. Then use the object chooser to choose the particular object to be the starting element. For more help on the object choose see the [Find Dialog Basics](#).

### Distance Measure

Choose the distance measure that will be used to determine distances between nodes. Each branch will be treated as having a length based on the choice below. Note: negative values are not allowed, therefore negative values will be treated as extremely small lengths instead.

- X the per unit series reactance.
- |Z| the magnitude of the series impedance (based on the per unit series reactance and resistance)
- Length the Length Field for each branch
- Number of Nodes Length of 1.0 is used for all branches
- Other When choosing Other, click the Find.. button to choose any numeric field of a branch

### Lines to Process

Regardless of the Distance Measure above, you can choose which branches allowed to be traversed when finding the shortest path.

- All All branches are allowed to be traversed
- Only Closed Only branches that are presently closed can be traversed
- Filter Only branches that meet the [Advanced Filter](#) specified can be traversed . Click **Define...** to choose or create an advanced filter.
- Selected Only branches whose Selected? field is set to YES can be traversed

### Bus Field to Populate

Choose a field into which the resulting path distances will be copied. It is recommended that you use the custom floating point or integer fields for this purpose.

---

## Closet Facilities to Point

---

To produce a list of display objects and their distance from a point selected on the one-line diagram, go to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **GIS Tools > Closest Facilities to Point**. After selecting this option, the mouse cursor will become a crosshair. Move the cursor to a location on the diagram and left-click. A dialog will open listing all of the display objects and their distance from the selected point in ascending distance from the point.

### Select to Show Display Objects of Type

This allows the option of showing all display objects in one list or only listing objects of a particular type.

### Distance

This field is listed by default with the display objects. This shows the distance between a given display object and the selected point. The location of the selected point is given by the **X/Longitude** and **Y/Latitude** fields on the dialog. The location of a given object is provided by the **X/Longitude Location** and **Y/Latitude Location** fields with the display object. The units of the distance is given in the Units field on the dialog.

### X/Longitude and Y/Latitude

Each display object listed has a corresponding **X/Longitude Location** and **Y/Latitude Location** field. The **X/Longitude** and **Y/Latitude** fields given on the dialog provide the location of the point that was selected on the diagram. The fields on the dialog can also be manually adjusted to determine the distance between the display objects and a given coordinate.

All of the fields that give the location of either the display object or the point selected on the diagram are given in x,y coordinates if a valid map projection is not selected or the option to show coordinates in longitude, latitude is not selected. Otherwise, the coordinates will be longitude, latitude. Both of these options can be set from the Online Display Options dialog in the [Geography/Coordinates](#) category.

### Units

This indicates the units that the distance is in. The units will be in PowerWorld units if a valid map projection is not selected or the option to show coordinates in longitude, latitude is not selected. Otherwise, the units will be in either miles or kilometers. To display the distance in miles, the option to use the **English Measurement System** must be selected on the PowerWorld Simulator Options dialog under the [Environment](#) category. Select **Metric (SI)** as the Measurement System to display the distance in kilometers.

### Recalculate Distances

Click this button to recalculate distances after changing the coordinates of the selected point by entering new values for the X/Longitude or Y/Latitude fields.

---

## Online Screen Coordinates

---

Onelines can be any size and can contain any number of objects. The size and position of objects on the screen are specified in terms of x-y "online screen coordinates." When the zoom level is 100% the size of the online in screen coordinates is 100 by 100. More generally, the size of the online is  $100^2$  divided by the zoom level in both the x and y directions.

The default screen center is the point (50,50) but this can be easily changed (See [Online Panning and Zooming](#).) Negative screen coordinates are allowed. Usually you will not have to be concerned about an object's location in screen coordinates.

The [Status Bar](#) displays the current cursor screen coordinates while in Edit Mode.

Edit Mode	X = 0.11 Y = 31.33
-----------	--------------------

**Status Bar Showing Screen Coordinates**

---

## Online Zooming and Panning

---

All online diagrams permit zooming and panning. Zooming and panning are very important tools for building and investigating large online diagrams that occupy more than a single screen of viewing area. Moreover, the display of various objects can be set to depend on the zoom level. See [Online Conditional Display of Objects](#) for more information on this feature.

The following mechanisms are provided for zooming or panning a oneline:

### Zooming on Onelines

Zooming involves adjusting the oneline diagram's display area by changing the magnification of the view. Zoom in on a oneline to have the screen display less of the complete oneline diagram, and zoom out on a oneline to have the screen display more of the complete oneline diagram.

To zoom on a oneline diagram using the keyboard, follow these instructions:

- Use CTRL-up arrow to zoom in
- Use CTRL-page up to zoom in quickly
- Use CTRL-down arrow to zoom out
- Use CTRL-page down to zoom out quickly
- The [Zoom](#) ribbon group offers additional zooming options:
- Use the **Zoom in on Area** button of the [Zoom](#) ribbon group to select a region on which to zoom.
- Use the **Show Full** button of the [Zoom](#) ribbon group to zoom the display out to show the entire oneline.
- Use the **Find** button of the [Zoom](#) ribbon group to display the [Pan/Zoom Dialog](#).
- Use the **Save View** button of the [Active](#) ribbon group to display the [Save View Dialog](#), or click on the drop down arrow on the Save View button to choose from a list of saved views.
- Use the **Rectangular Zoom Selector** to select a section of the diagram to zoom into using a selection box. After clicking button (symbolized with a magnifying glass over a dashed rectangle,) left click and hold the mouse button down on the diagram, drag the mouse to select part of the diagram inside a box, then release the mouse button. Simulator will zoom into the region selected inside the box.

### Panning on onelines

Panning moves the screen's focus point around the oneline diagram. You can pan left, right, up, or down to view different portions of the complete oneline diagram.

To pan around the oneline using the keyboard, follow these instructions:

- Use the arrow keys to move in the desired direction
- Use page up to move up quickly
- Use page down to move down quickly
- Use the Home key to move left quickly
- Use the End key to move right quickly
- Left-click and hold the mouse button down anywhere on the *background* of the diagram. Make sure you do not click on an object on the diagram. While holding the left mouse button down, drag the mouse in any direction to "drag" the diagram in that direction.

You can also use the [Pan/Zoom Dialog](#) to pan to a specific location, or even to a specific bus. You can display the Pan/Zoom Dialog either by pressing the **Find** button on the Zoom

ribbon group of the Onelines ribbon tab or by selecting **Pan/Zoom Control** from the online diagram's [local menu](#).

---

## Save View Level Dialog

---

The **Save View Level Dialog** is used to save an (x,y) location and zoom level, contour description, and/or hidden layer application for a online diagram in an easily accessible list for quick recall and application to the diagram. This dialog can be called by selecting the **Save/Edit/Delete View** option from the [online local menu](#).

You can also define a new layer by going to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **Save View > Save View**.

Recalling a saved view can be done by either right-clicking on the background to access the [online local menu](#) and choosing a view from the **Go To View** list, or by clicking on the drop down arrow on the **Save View** button and selecting a view from the resulting list. Selecting a saved view automatically moves the online diagram location to the (x,y) coordinates and zoom level, applies a stored contour, and/or applies hidden layers stored with the selected view.

This dialog can be used for creating a new view, editing an existing view, or deleting an existing view. The **Save View Level Dialog** has the following options:

### View Name

This is the name that the view will be stored under in the saved view list. This must be a unique name for each view saved with the online diagram. By default this field is blank. To edit an existing view, choose the desired view from the drop down list.

### Save Display Information in View

When checked, the view will store an x-y coordinate and zoom level to associate with the view. When the view is selected for display, the diagram will center on the x-y coordinate at the defined zoom level.

### X-Coordinate

The x-coordinate for the view. This value will default to the current x-coordinate for the online diagram, or will change to a saved value if a saved view is chosen from the **View Name** drop down list. This value can be modified by the user.

### Y-Coordinate

The y-coordinate for the view. This value will default to the current y-coordinate for the online diagram, or will change to a saved value if a saved view is chosen from the **View Name** drop down list. This value can be modified by the user.

### Zoom Level

The zoom level for the view. This value will default to the current zoom level for the online diagram, or will change to a saved value if a saved view is chosen from the **View Name** drop down list. This value can be modified by the user.

### Save Contour Information in View

If a contour is being displayed when the view is created, you can choose to save the [contour](#) information with the view by checking this box. Saving a contour with a view will display that contour when you switch to that view. You also have the option of saving a blank contour with the view. This is indicated by a Contour Object of 'None' and a Contour Field of 'None.' If you save a blank contour, no contour will be displayed when switching to that view. If you don't save any contour with the view, then any existing contour will be applied when switching to that view.

### Contour Object

This section displays the type of object the stored contour pertains to. This field cannot be changed from this location.

#### **Contour Field**

Displays the type of value the stored contour pertains to. This field cannot be changed from this location.

#### **Link to Online Display Options Settings**

In the [Online Display Options](#) in Simulator, you can save various sets of options with a case for quickly recalling online diagram settings by name. This option allows you to include a custom defined option set by name with the current view. Note that you first must have created and saved a custom set of options for a diagram from the [Online Display Options](#) dialog before this option on the Save Views dialog will be enabled.

#### **Save Hidden Layers**

Since Simulator allows the use of [layers](#) to display or hide objects on the diagram, views also can optionally store the layer settings when the view is created. The window below this checkbox will list the layers that are presently hidden on the diagram. When this view is recalled, the layers in this list will be hidden if they were stored with the view.

#### **Save**

This button will save a new view or modify the values for an existing view of the name in the **View Name** field.

#### **Delete**

This button will delete the currently open view from the saved view list. The dialog information will default to the saved view information of the previous view in the list.

#### **OK**

This button will save a new view or modify the values of an existing view of the name in the **View Name** field, and will close the **Save View Level Dialog**.

#### **Save to AXD**

By clicking this button, all the views will be saved to a [display auxiliary file](#).

#### **Load from AXD**

This button will load all the views saved in a [display auxiliary file](#).

---

## Window Ribbon Tab

---

The Window ribbon consists of buttons and menus that provide access to customizing the windows in the user interface. It also provides access to help topics and topics regarding the Auxiliary File Format. The ribbon is shown below.



The Case Information ribbon is broken into four ribbon groups

### Mode Ribbon Group

#### Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

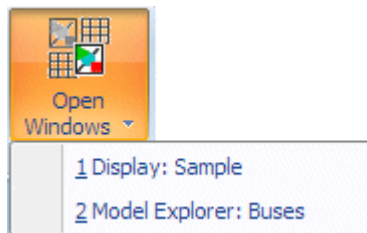
#### Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

### Window Ribbon Group

#### Open Windows Menu

The Open Windows Menu provides a convenient method of switching between open windows. This includes User Interface Dialogs, Case Information Displays, and Online diagrams.



#### Switch to Free-Floating Windows

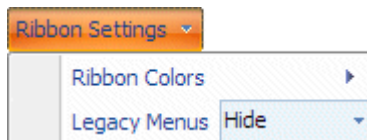
Choosing this option toggles Simulator from containing all diagrams within the Simulator program shell to all windows related to Simulator being free-floating on your display.

#### Refresh Displays

Redraws (refreshes) each of the open windows. Simulator usually automatically refreshes the open windows as necessary. However, this action allows you to trigger the refresh when you want it.

#### Ribbon Settings Menu

This menu contains options for customizing your ribbon.





**Ribbon Colors** : You can change the color scheme to either **Blue**, **Silver** or **Black**

**Legacy Menus** : Choose to Hide or Show a special Menus ribbon tab. The Menus ribbon tab mimics the menu and toolbar structure that existed in older versions of Simulator.

#### **Tile**

Rearranges the open online diagrams such that the total window area is divided equally among all of them and each is completely visible.

#### **Cascade**

Rearrange all open windows such that all they appear on top of each other while leaving the title bars visible.

#### **Toggle Full Screen**

Toggle full screen makes the currently selected diagram window switch to full screen mode. Full screen mode will dedicate the screen to the window, with all other windows, including the Simulator menus and toolbars, to be hidden. To get back to normal mode, right-click on the diagram in full screen mode and select Toggle Full Screen from the popup menu.

### **Help Ribbon Group**

#### **Contents**

Opens the [Simulator Options Dialog](#).

#### **Set Help File**

Opens the [Case Description Dialog](#).

#### **About**

Click this to open the About Dialog. The About dialog contains information about the software as well as showing you the Build Data of your version of Simulator. PowerWorld release patch version of Simulator on our website. These patch versions are named by date and is shown on the About Dialog.

#### **PowerWorld Website**

Click this to open your default web browser to the PowerWorld Corporation Website.

#### **Check for Updates**

This button will open the PowerWorld website to the patch download page. You will be required to enter your Account Username and Product ID (password) to check for and download patches for your version of Simulator.

### **Auxiliary File Ribbon Group**

#### **Load Auxiliary**

Choose this option to load an [Auxiliary File](#).

#### **Load Display Auxiliary File**

Choose this option to load a [Display Auxiliary File](#) to append to the active online.

#### **Auxiliary File Format**

Opens the [Bus View Online](#) which provides a graphical display which allows you to quickly browse information about a bus and everything connect to that bus.

#### **Export Display Object Fields Menu and Export Case Object Fields Menu**

Choose these options to export a list of fields for each type of object in a case (or a Online Display). The list also indicates which fields are [key fields](#) and [required fields](#) for each object. You can output this list of fields as either a text file or into Excel.



---

**Online Conditional Display of Objects**

---

Along with supporting zooming and panning, the onelines permit the conditional display of objects. That is, it is possible to specify display objects so they are visible only at particular zooming ranges. This enables the oneline to show additional details as the user zooms in and fewer details when the user zooms out.

Please note that the zoom levels are defined as percentages. If you want an object to display only between 50% and 150%, you must select 50 and 150 as the zoom level boundaries.

This option is available by assigning objects on the diagram to a [Layer](#). Layers can be defined and set to objects on the [Format Multiple Objects](#) dialog.

---

## Keyboard Short Cut Actions Dialog

---

The Keyboard Short Cut Actions dialog allows the user to associate a keyboard shortcut to a determined oneline diagram, and even to a particular [view](#) of that diagram. When the keyboard shortcut is pressed, the oneline diagram is opened (if it was closed) and brought to the front. When a view is also associated, then the user is taken to such a [view](#).

To open the Keyboard Short Cut Actions Dialog, select **Keyboard Shortcuts** on the **General Options** ribbon group of the [Options](#) ribbon tab. These General Options are also available on the [Oneline](#) ribbon tab.

The dialog has the following controls:

### Keyboard Shortcuts

Lists all the keyboard shortcuts. When selecting a keyboard shortcut, it will show the *oneline* and the *view* associated to the selected keyboard shortcut. If your keyboard has an extra row of function keys, then you can check the box labeled **Include F13-F24**. If a shortcut has been defined for the respective key, then it will be highlighted and a string will summarize the action of the shortcut

### Clear All

Clicking this button will delete all the associations of all the listed shortcuts.

### Save Shortcut

Click this button after entering a oneline diagram and eventually a view so they can be associated with the selected keyboard shortcut.

### Delete Shortcut

Click this button to delete the associations of the selected keyboard shortcut with any oneline diagram and view.

### Create Shortcut

This indicates the keyboard shortcut to which the below actions will be assigned. The available actions are to Open a Oneline and to Open an Auxiliary File

### Open Oneline

The following options are available for opening oneline actions:

#### Oneline and View: Oneline

Shows the path and the name of the oneline diagram which will be opened when the selected keyboard shortcut is pressed. If the field is blank, then no oneline diagram is associated to the keyboard shortcut. You can click **Browse** to find the oneline diagram.

#### Oneline and View: View

Shows the name of the view belonging to the associated oneline diagram to which the user will be taken when the selected keyboard shortcut is pressed. If the field is blank, then no view is associated to the keyboard shortcut.

### Action for already open onelines

If the oneline being requested by the shortcut key is already open, then this option determines what should be done. Choices are to close the oneline, bring the oneline to the front, or to open a new oneline.

### Where to open oneline

When opening a oneline via a shortcut key, you may do one of the following:

- Separate Window (open the oneline as normal)

- Separate Window – [Toggle Full Screen](#) (open normal and then toggle it to full screen)
- Embed the Active Online Window (opens the online and embeds it inside the presently active online diagram)

Note: if onelines already exist which have been toggled to full screen, and a short cut key is used to open a window that will not be toggled to full screen, then all other onelines present open will be switched to normal mode instead of full screen mode.

### Size and Location

This specifies the relative size and location you would like the online to be opened at. When opening the online normally, the percentage is relative to the size of the container window. When opening the online embedded in the active online, the percentage is relative to the size of the active online.

### Make embedded online borderless

When [embedding](#) the online inside the presently active online, this option will make the embedded online borderless.

### Open Auxiliary File

The following options are available for opening auxiliary file action:

#### Filename

Shows the path and the name of the auxiliary file which will be opened when the selected keyboard shortcut is pressed. If the field is blank, then no auxiliary file is associated to the keyboard shortcut. You can click **Browse** to find the auxiliary file.

#### Section name

Shows the name of the [script](#) or [data](#) section belonging to the associated auxiliary file which will be executed when the selected keyboard shortcut is pressed. If the field is blank, then no specific section is associated to the keyboard shortcut, so all script and data sections will be loaded.

### Create objects if they do not already exist

Check this box to indicate that new objects in the auxiliary file should automatically be created.

### Save/Load shortcuts to/from an Auxiliary File

Click the **Save** button to save all the shortcuts to an auxiliary file. Click the **Load** button to load the shortcuts saved in an auxiliary file. The shortcuts saved in the auxiliary file will replace the shortcuts saved in the dialog.

### Close

Closes this dialog.

In addition to the shortcuts displayed on this dialog, Simulator also offers standard windows editing shortcuts such as:

**Ctrl + x** – Cut Command

**Ctrl + c** – Copy Command

**Ctrl + v** – Paste Command

---

## Printing Online Diagrams

---

To print a online diagram, select **Print Online** from the [Application Button](#) or right-click on the online background and select **Print Window** from the popup menu. This opens the [Print Options Dialog](#), which you can use to configure the print job, including its size, orientation, border, and title bar. You can even choose to print a online to multiple pages using Simulator's multi-page printing.

---

## Print Options Dialog

---

The Print Options Dialog is used to configure the printing of online diagrams (including bus view displays) and of strip charts. To print an online diagram, select **Print Online** from the [Application Button](#). To print a strip chart, right-click on it to invoke its local menu, and then select **Print Window**. In both cases, Simulator will open the Print Options Dialog.

The Print Options Dialog has three tabs:

### Page Layout

This tab sheet contains options for how the diagram will appear on the printed page. It presents the following controls:

#### Margins

Specify the horizontal and vertical margins in either inches or centimeters.

#### Scaling

Set this option to **Proportional** to print the online diagram or strip chart such that the screen's aspect ratio is maintained. Set this option to **Fit to Page** to force the printout to take up all available space on the page in both horizontal and vertical directions.

#### Draw Border

Places a border to be drawn around the online diagram or strip chart.

The second tab controls the printing of a title bar. Check the **Print Title Bar** box to print a title bar at the bottom of the diagram. Checking this box displays a number of options that allow you to specify various items to include in the title bar. If you don't want to include certain items on the plot, simply leave those fields blank.

The title bar is split horizontally into three sections. If you specify values for all requested items, they will be arranged in the title bar as follows:

Company | Description Line 1 | Date  
 Department | Description Line 2 | Drawing #  
 Author | Description Line 3 | Title

### Title Bar

This tab is used to specify whether or not to include a title bar on the diagram, along with the information that would be contained in the title bar.

### Multi-Page

This tab allows the online diagram to be "divided" into a specified number of sections for printing the online to several pages. A multi-page online can be useful for including "close-up" views of areas of a diagram in a report, or the multiple pages can be cut and combined to form a larger printed online if a plotter is not available. The **Multi-Page** tab presents the following controls:

#### Grid Size

Choose this value to subdivide the online into an N by N grid. You can then choose which sections of the online to print based on the grid overlay. The maximum size is 10 x 10.

### Choosing What to Print

A low resolution picture of your oneline is shown on the **Multi-Page** tab. Gridlines are drawn over the diagram to show where the oneline has been subdivided based on the **Grid Size** chosen.

To remove a grid section from the printout, click on the oneline image in the corresponding section of the grid. The selected section of the grid will become dark to indicate you wish to prevent the section from printing.

To add a grid section back into the print job, click on a darkened section of the grid to allow that section of the oneline to be printed.

The number of pages you have chosen to print will be shown on the dialog. When printing out a multi-page oneline with a title bar, the page number will appear on the title bar in the form "Column Letter"-"Row Number". For example, if the Grid Size is 4, the pages are numbered **A-1** through **D-4**.

Click **Print** to send the document to the printer, or **Cancel** to abort the print. Click **Setup** to view the default Windows printer dialog, which will allow you to specify whether to print the figure in portrait or landscape modes and to set various printer-specific properties.



---

**Printer Setup**

---

Choose **Printer Setup** from the [Application Button](#) to configure the printer using the standard Windows printer setup dialog. This dialog allows you to define which printer to use for printing from Simulator, the size of the printed page, the page's orientation, and additional properties that are specific to the printer you are using.

---

### **Chapter 3: Creating, Loading, and Saving Simulator Cases**

---

This chapter describes the File and Simulation Control Menus. The File Menu is used to open and save either full cases or oneline diagrams, or to create new cases or new onelines (Edit Mode). The menu is also used to exit Simulator or to validate a case (Edit Mode).

The Simulation Control Menu controls the actual simulation. You can use this menu to do either a time-domain simulation or a single Power Flow Solution.

The following material is included:

- File Menu
- Case Formats
- Validating a Case
- Opening Files
- Creating Files
- Closing Files
- Additional File Formats

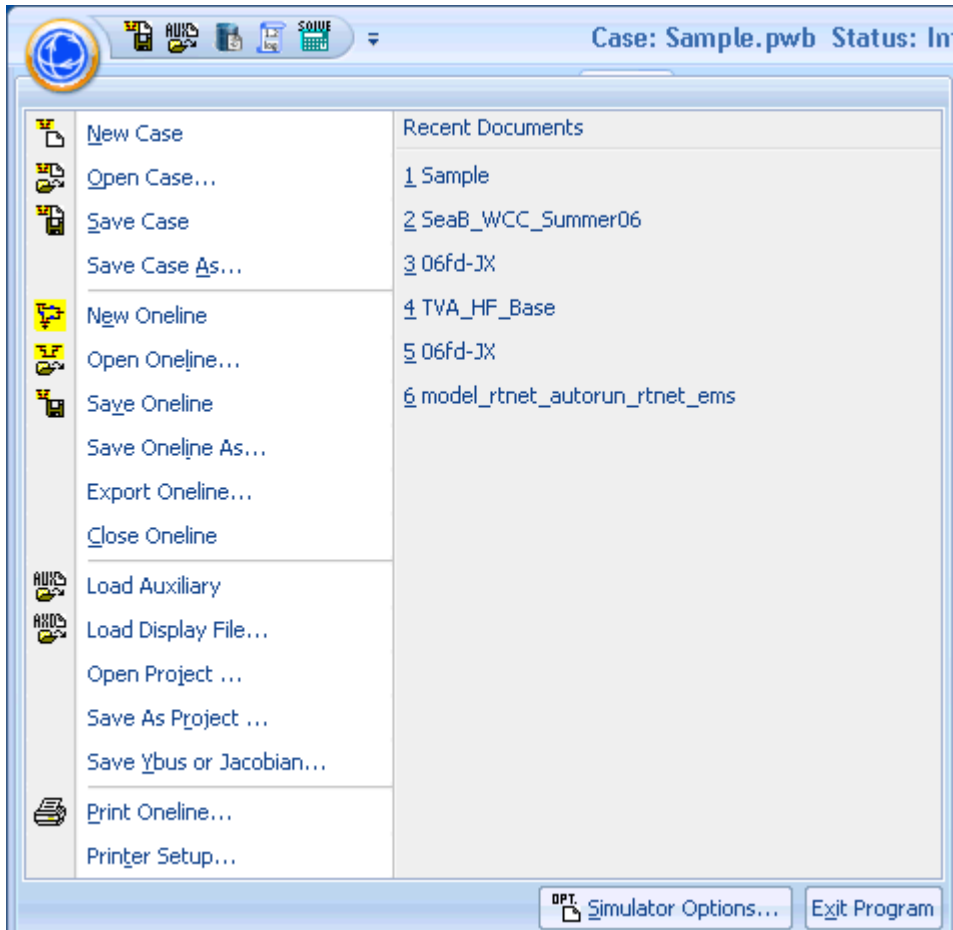
---

## Application File Menu

---

The Application File menu is used to open and save either full cases or oneline diagrams, to create new cases, to open and load auxiliary files, to print and to exit Simulator. To access this menu you must click on the Application button in the upper-left corner of the program.

Application Button:



---

## Case Formats

---

Simulator supports a number of case formats. You can both open and save cases using most of the following formats:

### PowerWorld Binary (\*.pwb) (Preferred Format)

For most users, the best choice of power flow case formats is the PowerWorld Binary format. This format stores the most complete set of case information but requires the smallest file sizes. Information stored in this format includes power flow data, economic parameters, case time variation/options values, and screen customizations. The only potential disadvantage of this format is that it is stored in binary form, which means that it cannot be viewed using a standard text editor. All the other formats are ASCII and thus readable in text editors such as Notepad.

One additional benefit of using the Powerworld Binary format is that with this format the case can be password-protected. This setting is available under **Misc. Power Flow > Set Case Username and Password** in the **Case Options** ribbon group on the [Options](#) ribbon tab.

### PowerWorld Auxiliary File (\*.aux)

Simulator can now save and load cases using the PowerWorld defined auxiliary file format. When the aux file format is chosen as the save file type in Simulator, the data describing the entire case is written to a text file described by Simulator's auxiliary file format. These files can be considerably larger than PowerWorld Binary files, and generally should not be used unless otherwise necessary. Note that not all auxiliary files represent an entire case, and in fact in most cases do not. Only when an auxiliary file is created using the **Save Case As** option in the [Application Button](#) in Simulator does the auxiliary file contain all necessary information for loading the complete case in Simulator as an auxiliary file.

### PTI Raw Data Format (\*.raw)

This format is included primarily for interchange of power flow data with other packages. The PTI Raw Data format only contains power flow data. When using this format, the generator participation factors are automatically set to be proportional to the MVA rating of each generator.

### PTI Raw Data Format (\*.raw) (with options)

This format is only available when opening a case. This format is the same as the PTI Raw Data Format except that a dialog will open allowing user input on how various options should be set. There is currently only one option to set. This option allows the user to select if transactions should be created for loads that are in different areas than their terminal buses.

Loads may be assigned to a different area than the load's terminal bus. PTI offers an option that allows the user to ignore this assignment when calculating the tie-line flow for an area.

(AREA INT CODE = 1 FOR LINES ONLY)

(AREA INT CODE = 2 FOR LINES AND LOADS)

Simulator does not allow you to define a load this way and then choose to ignore it. Most RAW files that PowerWorld has seen seem to be solved ignoring the loads that are in different areas than their terminal bus by using the "lines only" option. To overcome this, when PowerWorld reads a RAW file, [MW Transactions](#) can be automatically created with the ID "RAW\_LOAD". These transactions are between the load's area and its terminal bus and are created so that the export from each area correctly matches a case solved using "lines

only". If you know that your case was solved using "lines and loads", then do not choose to add the transactions when prompted. See the topic **Add and Remove transactions due to tie-line loads** with [File Management Options](#) for more information about adding or removing these transactions after the case has been loaded.

If not loading the RAW file with options, the transactions representing loads that are in different areas than their terminal buses will be automatically created.

#### **GE PSLF Format (\*.epc)**

This format is included primarily for importing load flow data saved from GE's PSLF program. The EPC format is a text file format. This format does contain excess information that Simulator does not use, however Simulator does store this information for editing and writing back out to EPC format files.

#### **GE PSLF Format (\*.epc) (with options)**

This format is the same as the GE PSLF Format except that the [GE EPC File Load Options](#) dialog will be opened to allow user input for various options.

#### **IEEE Common Format (\*.cf)**

The IEEE Common Format is used to specify only power flow information. CAUTION: IEEE common format does not support many of the formats used in the PowerWorld packages, such as multiple loads and generators at a bus. Usually, IEEE Common Format is used only for inputting cases.

#### **PowerWorld Simulator Project Files (\*.pwp)**

Project files are described in [Overview of PowerWorld Simulator Project Files](#).

Advanced settings for opening cases are available under the [File Management settings](#).

## GE EPC File Load Options

The GE EPC File Load Options dialog will open when choosing to open or save a GE EPC case "with options".

**GE EPC File Load Options**

File  
EPC File Version: Version 15

What To Do With Multi-Section Lines  
☒ Maintain sections  
☐ Equivalence sections when possible

Assumed Generator Var Limit Deadband: 2.000  
 Note: if [min..max] var range is smaller than this, then Simulator will assume AVR = NO

☐ if GE Base Load Flag > 0, then Post-CTG Prevent Response = YES, else Post-CTG Prevent Response = NO

Filter By...

☒ Area/Zone/Owner Filters ☒ Include Tie Lines  
 Enter numbers separated by commas (leave blank for all)

Areas:   
 Zones:   
 Owners:

OK Cancel Help

### EPC File Version

Set this to determine what version of the EPC file is being read or saved.

### What To Do With Multi-Section Lines

### Assumed Generator Var Limit Deadband

### GE Base Load Flag Setting

When opening a file, the GE Base Load Flag setting will be used to set the Post-CTG Prevent AGC Response field in Simulator. The caption will be the following: **if GE Base Load Flag > 0, then Post-CTG Prevent Response = YES, else Post-CTG Prevent Response = NO**

When saving a file, the Post-CTG Prevent AGC Response field in Simulator will be used to set the Base Load Flag setting in the EPC file. The caption will be the following: **if Post-CTG Prevent Response = YES, then set GE Base Load Flag = 1, else set GE Base Load Flag = 0**

### Filter By

---

**Validating a Case**

---

Validation checks for both errors that would prevent a case from working properly in Simulator and for abnormal setup of model objects. Validation errors and warnings are displayed in the [Message Log](#). A sampling of the tasks performed during case validation include:

- Ensures there is a system slack bus assigned
- Ensures the slack bus has an attached generator
- Ensures that no generator is assigned to remotely regulate the slack bus
- Checks for buses having multiple controllable devices attached (i.e. switched shunts)
- Checks all max/min enterable fields to ensure that the max value is greater than the min value

Case validation is performed automatically when switching from Edit Mode to Run Mode. If a case has validation errors, Simulator will not allow the user to switch to Run Mode. Warnings will not prevent switching to Run Mode.

---

## Opening a Simulation Case

---

The most common first step in using Simulator is to open a case. When a case is opened, any associated oneline files are also opened. To open a case, either select it from the [list of recently opened cases](#) in the file menu or

- Select **Open Case** from the [Application Button](#) to display the Open Dialog.
- In the **Type of Files** box, select the desired file type. By default, the PowerWorld Binary type is selected (\*.PWB). The PowerWorld Binary is the preferred file type, providing the most comprehensive power system information along with the smallest size and quickest load time. Other file types include the PowerWorld Case type, PowerWorld Auxiliary file, PTI Raw Data formats, GE EPC text format, and IEEE Common Format. Please see [Case Formats](#) for more details.
- In the list of cases, click on the desired case.
- Click OK.

When opening an existing Simulator case, you may see one or more oneline diagrams pop up. When opening a power flow case created with another program, a oneline may not be available. However, you can easily create a oneline for such a case using the [Edit Mode](#).

If you are in the Edit Mode, you can modify the case. See [Edit Mode Overview](#) for details.

If you are in the Run Mode, the Clock Window may be displayed. This window shows the simulation start time, end time and current time. You may choose to close or open the clock at any time.

If the case has validation errors, the mode is immediately switched to Edit Mode. You must correct the validation errors shown in the message log before you can use the Run Mode to solve the case.



---

## Opening a Oneline Diagram

---

Simulator supports oneline diagrams that have been developed using its Edit Mode tools. PowerWorld can also import PTI Draw Files (\*.draw).

When a case is opened, any associated onelines are usually opened, as well. Additionally, oneline diagrams can be opened directly from any existing onelines using [oneline links](#). However, you may also directly open a oneline diagram using the following procedure:

- Select **Open Oneline** from the [Application Button](#).
- Choose the file type you would like to open (One Line Display File, PTI Draw File, or [Display Auxiliary File](#)).
- In the list of available oneline files, click on the desired oneline.
- Click OK.

You may open as many onelines as you like, and even multiple copies of the same oneline.

The following topics may be helpful when opening onelines created with a different case:

[Refresh Anchors](#)

[Bus Renumbering Dialog](#)

---

**Recently Opened Cases**

---

A numbered list of the most recently opened cases appears in the Recent Files list when you click on the [Application Button](#). Simply choose a case from the list to open the case.

---

## Building a New Case

---

To create a new case in Simulator, select **New Case** from the [application file menu](#). After asking you whether or not to save the current working case (if one exists), Simulator will automatically switch to Edit Mode. The screen will turn to the default background color, indicating that you can begin to build the new case.

New users may wish to view the tutorial [Creating a New Case from Scratch](#) for further guidance.

---

## Building a New Online

---

This option enables you to create a new online diagram. It is accessible from either the Edit or Run modes by choosing **New Online** from the [application file menu](#). When you select it, the application will automatically switch to Edit Mode, from which you can construct the new online.

New users may wish to view the tutorial [Creating a New Case from Scratch](#) for further guidance. Additionally, the following topics may be of some assistance:

[Using the Insert Palettes](#)

[Online Conditional Display of Objects](#)

[Using the Online Alignment Grid](#)

---

## Saving Cases

---

Simulator allows users to save case information in several different formats. To call up the Save As dialog, select **Save As** from the [Application Button](#). The format in which the file will be saved depends upon the value of the **Save as Type** field of the dialog. Simulator can save cases in PowerWorld binary format (the default), PowerWorld Auxiliary file format, PTI raw data, GE PSLF format, and the IEEE common format. Please see [Case Formats](#) for more details on the various case formats available.

To convert a case to a different format, follow this procedure:

- Select **Open Case** from the [Application Button](#).
- In the **Files of Type** box, select the desired file type. By default, the PowerWorld Binary type is selected (\*.PWB).
- Select the name of the desired case and click *OK*.
- Select **Save Case As** from the [Application Button](#).
- Change the **Save as Type** setting to match the desired file type, designate the name with which to save the file, and click *OK*.

To simply save the case with its current name and format, select either **Save** from the [Application Button](#) or the **Save Case** button on the Quick Access toolbar.

---

## **Saving a Oonline**

---

Select **Save Oonline** from the [Application Button](#) to save the currently selected oonline file.

Select **Save Oonline As** from the [Application Button](#) to save the currently selected oonline file with a different name or format. You may overwrite existing files.

For both the Save Oonline and Save Oonline As options, only the oonline diagram is saved; the case (i.e. the power flow model) is not written to disk.

---

**Close Online**

---

To close a online diagram, select to **Close Online** from the [Application Button](#), or right-click on the background of the online diagram and select **Form Control > Close** from the local menu. This will prompt you to save the online if changes have been made since the last save. Note that this does not close the current case, just the online diagram. The power system data is still loaded, and case simulations can be run with the online closed.

---

### **Auxiliary File Format (\*.aux)**

---

Simulator Auxiliary Files have now been greatly enhanced to include both Script sections for running Simulator commands in a "batch" process, and Data sections for changing mass amounts of load flow information. Many [auxiliary file formats](#) used in early versions of Simulator can not be saved in current versions. The older formats can still be read into Simulator.

Auxiliary file format and use now has its own help chapter titled [Auxiliary Script/Data Files](#).



---

**Generator Capability Curves Format (\*.gcp)**


---

These files are outdated for Simulator version 8.0 and later. Their descriptions are still maintained here as they can still be read into Simulator. However, they can no longer be saved. These records are now stored in [Auxiliary Script/Data Files](#).

The generator capability curves are used to model the dependence of the generator reactive power limits and the generator's real power output. The generator reactive capability curves files have the following format, with one generator per line:

**num or name, ID, P<sub>1</sub>, Q<sub>1,max</sub>, Q<sub>1,min</sub>, P<sub>2</sub>, Q<sub>2,max</sub>, Q<sub>2,min</sub> ... P<sub>n</sub>, Q<sub>n,max</sub>, Q<sub>n,min</sub>**

where

**num/name** the generator's bus number or the bus' name in single quotes,

**ID** the generator's single character id,

**P<sub>i</sub>** a generator MW output value,

**Q<sub>i,max</sub> and Q<sub>i,min</sub>** the associated maximum and minimum reactive power limits.

Version 11 and later of Simulator allow for an unlimited number of capability curve points to be read from the gcp file. Up to 10 different P/Q<sub>max</sub>/Q<sub>min</sub> values may be specified in older versions of Simulator.

---

## Generator Cost Data Format (\*.gcd)

---

These files are outdated for Simulator version 8.0 and later. Their descriptions are still maintained here as they can still be read into Simulator. However, they can no longer be saved. These records are now stored in [Auxiliary Script/Data Files](#).

The generator cost data file contains the parameters used to model the generator operating costs. Each record in the generator cost data file specifies the operating cost model for a single generator. The cost model may be either a cubic polynomial or a piecewise linear curve. When reading a file of generator cost data records, Simulator will open the [Cost Curve Data Options Dialog](#). This dialog gives you additional control over the input and process of generator cost information.

Each generator cost data record begins with the fields

**num or name, ID,**

where

**num/name** the generator's bus number or the bus' name in single quotes,

**ID** the generator's single character id,

Following the ID field is an optional single-character field **CostModel** that specifies the type of cost model. If **CostModel** is 'CUBIC' or is not specified, the record describes a cubic cost model and thus concludes with the following fields:

**a<sub>i</sub>, b<sub>i</sub>, c<sub>i</sub>, d<sub>i</sub>, FuelCost ParFac**

where

**a<sub>i</sub>, b<sub>i</sub>, c<sub>i</sub>, d<sub>i</sub>** input-output curve coefficients; see [Generator Cost Information](#) for details,

**fuelcost** fuel cost, expressed in price per Mbtu,

**ParFac** generator participation factor.

If, on the other hand, **CostModel** is 'PLIN', the record describes a piecewise linear cost model. The remainder of the record specifies pairs of generator output and corresponding generator cost:

**FixedCost MW<sub>1</sub> IncCost<sub>1</sub> MW<sub>2</sub> IncCost<sub>2</sub> ... MW<sub>n</sub> IncCost<sub>n</sub> MWneg ParFac**

where

**FixedCost** Operation cost independent of the generator's MW output; expressed in \$/Hr,

**MW<sub>i</sub>, IncCost<sub>i</sub>** Output/incremental cost data point pairs that define the incremental cost model for the generator (note IncCost<sub>i</sub> has the units \$/MWhr),

**MWneg** An arbitrary negative value used to terminate the record,

**ParFac** generator participation factor.

---

## Generator Cost Curve Data Options

---

The **Cost Curve Data Options Dialog** gives you additional control over the input and processing of generator cost information. It appears when you try to read generator cost information from an [auxiliary data file](#) (either [.aux](#) or [.gcd](#)).

To command Simulator to turn all units onto automatic generation control (AGC) for which it reads a cost data record, check the first checkbox. This box has the rather lengthy label "If a unit for which a cost curve is read is not on AGC, Simulator should set it on AGC." In other words, if a cost curve is read for a particular generator, that generator will be set on AGC if this box is checked.

Cross-compound is a generator architecture sporting a single boiler and two turbines, one operating at high pressure and the other operating at low pressure. The existence of cross-compound units may complicate the modeling of generator cost characteristics, because sources for this information tend to describe the cost associated with the tandem, whereas Simulator's economic dispatch will try to dispatch the two parts of the cross-compound unit as two separate units. To address this problem, Simulator gives you the option to lump the properties of the high- and low-pressure turbines into a single unit. It consolidates the two components by adding together their maximum MW outputs, minimum MW outputs, and present output levels, assigning these quantities to one of the units, and setting the other unit off AGC and at 0 MW of output. To have Simulator perform this function for you, check the box in the *Cross Compound Units* panel.

To tell Simulator how to identify units that belong to the same cross-compound generator, click the box labeled "How To Identify...". This will open the [How to Identify Cross-Compound Units](#) dialog.

---

## Identify Cross Compound Units

---

Simulator regards cross-compound units as those that are connected to the same bus and that have generator id's that match any of a number of specified pairs. You must define for Simulator the pairs of generator id's that should be used to identify cross-compound units. Use the "**How to identify cross-compound units**" dialog to do this.

Specify the pair of generator id's in the small text boxes below the text labeled "Specify a new pair of id's that identify two units that belong to the same cross-compound set." Then, click the **Add** button to include the pair of id's you just entered in Simulator's cross-compound identification procedure. The box on the right lists all the pairs of identifiers Simulator will use to identify parts of a cross-compound set. To remove a pair of identifiers from this list, select it from the box and click the Delete key.

The list of cross-compound identifier pairs is stored in the pwrworld.ini file, so there is no need to specify these pairs each time you load in a new case.

---

**Injection Groups Format (\*.inj)**


---

These files are outdated for Simulator version 8.0 and later. Their descriptions are still maintained here as they can still be read into Simulator. However, they can no longer be saved. These records are now stored in [Auxiliary Script/Data Files](#).

The injection group files store the information for defined injection groups (also referred to as participation groups). The files have the following format:

**GROUP** *groupname*

**POINTS**

**Devtype** *busnum* *id* **participation** *pointtype*

.  
.  
.

**END**

where

**groupname** the name of the injection group, in single quotes,

**devtype** either GEN or LOAD,

**busnum** bus number of the device,

**id** the device's single character ID,

**participation** relative amount each device will contribute during a transfer,

**pointtype** either FIXED or DYNAMIC.

Multiple GROUP sections can be put in the file, each marked with END to signify the end of that group's data.

---

## Interface Data Format (\*.inf)

---

These files are outdated for Simulator version 8 and later. Their descriptions are still maintained here as they can still be read into Simulator. However, they can no longer be saved. These records are now stored in [Auxiliary Script/Data Files](#).

The interface data files store the interface data. The files have the following format:

```
INTERFACE intname lima limb limc  
Elemtype from to id checktoend fromto
```

```
.  
.  
.
```

where

**intname** the interface name, in single quotes,  
**lima, limb, limc** the three limits for the interface, in MW,  
**elemtype** one of three valid element types: AREA, LINE, or ZONE  
**from** for AREA and ZONE, the first area or zone number; for LINE, the from bus number,  
**to** for AREA and ZONE, the second area or zone number; for LINE, the to bus number,  
**id** only included for LINE, the two character circuit identifier; otherwise left blank,  
**checktoend** only included for LINE, if this field equals 1, the flow is checked at the to bus, otherwise the flow is checked at the from bus; otherwise left blank,  
**fromto** only included for LINE, if this field equals 1, the positive flow is assumed to be from the from bus to the to bus, otherwise the flow is assumed to be in the opposite direction; otherwise left blank.

Multiple interface records can be listed in the file, and multiple interface elements can be listed in each interface record.

---

## Sequence Data Format

---

The sequence data files store the sequence data needed to run a fault analysis. The sequence data format has the following five different types of records:

- Generator
- Load
- Branch
- Switched Shunt
- Mutual Impedance

Each of these four types of sequence data records has a different format, since they each store different information for the sequence data. The format of each record is given below:

### Generator

**GEN busnum ID Rpos Xpos Rneg Xneg Rzer Xzer RN XN**

where

**busnum** generator terminal bus,  
**id** generator identifier,  
**Rpos, Xpos** positive sequence resistance and reactance,  
**Rneg, Xneg** negative sequence resistance and reactance,  
**Rzer, Xzer** zero sequence resistance and reactance,  
**RN, XN** neutral-to-ground resistance and reactance.

### Load

**LOAD busnum Gneg Bneg Gzer Bzer**

where

**busnum** load terminal bus,  
**Gneg, Bneg** negative sequence total conductance and susceptance for all loads at bus,  
**Gzer, Bzer** zero sequence total conductance and susceptance for all loads at bus.

### Branch

**BRANCH fbusnum tbusnum ckt Rzer Xzer Czer fGzer fBzer tGzer tBzer Xftype**

where

**fbusnum, tbusnum** from and to bus numbers for the branch,  
**ckt** branch circuit identifier,  
**Rzer, Xzer** zero sequence branch resistance and reactance,  
**Czer** total zero sequence line charging,  
**fGzer, fBzer** zero sequence line shunt conductance and susceptance at the from bus end of the branch,  
**tGzer, tBzer** zero sequence line shunt conductance and susceptance at the to bus end of the branch,

**Xftype** Transformer configuration, entered as an integer based on the following table. Note that the default for a transmission line is 3, since a grounded wye - grounded wye transformer connection has the same equivalent model as a transmission line.

- 0 Wye - Wye
- 1 Grounded Wye - Wye
- 2 Wye - Grounded Wye
- 3 Grounded Wye - Grounded Wye
- 4 Wye - Delta
- 5 Delta - Wye
- 6 Grounded Wye - Delta
- 7 Delta - Grounded Wye
- 8 Delta - Delta

### Switched Shunt

**SSHUNT busnum numblocks Bzer1 Bzer2 ...**

where

**busnum** terminal bus number,

**numblocks** number of different zero sequence admittance blocks,

**Bzer#** zero sequence susceptance for each admittance block, maximum of 8.

### Mutual Impedance

**MUTIMP from1 to1 ckt1 from2 to2 ckt2 RM XM start1 end1 start2 end2**

where

**from1, to1** from and to bus numbers of the first mutually coupled branch,

**ckt1** circuit identifier of the first mutually coupled branch,

**from2, to2** from and to bus numbers of the second mutually coupled branch,

**ckt2** circuit identifier of the second mutually coupled branch,

**RM, XM** zero sequence mutual resistance and reactance,

**start1, end1** start and end locations of the section of the first line affected by the mutual coupling, with each point represented as a percentage of the total line length (between 0 and 1),

**start2, end2** start and end locations of the section of the second line affected by the mutual coupling, with each point represented as a percentage of the total line length (between 0 and 1).



---

## Exporting Onelines in Different Graphic Formats

---

Simulator can export oneline diagrams and other graphical displays as bitmaps, metafiles, or jpegs. Select **Export Oneline** from the [Application Button](#). A save file dialog appears where the dropdown menu for files type offers three choices: JPEG, BITMAP, METAFILE. Select the file type and choose the name before clicking the save button.

Exporting an image as a jpeg also requires you to set the compression ratio for the picture. See [Saving Images as Jpegs](#) for more information.

---

## Saving Images as Jpegs

---

Simulator can save oneline diagrams, bus view displays, and strip charts as jpeg images. To save a oneline diagram or bus view display as a jpeg, select **Export Oneline** from the [Application Button](#). This brings up the save file dialog. Choose jpeg as the file type, type the file name, and press save. This brings up the Jpeg Options Dialog where you decide the picture's resolution. Adjust the resolution control to specify the compression ratio at which to save the diagram as a jpeg. The greater the resolution you specify, the larger the resulting file will be. Click **Save** to save the image or click **Cancel** to terminate the process without saving the image as a jpeg.

To save a strip chart as a jpeg image, right-click on the background of the strip chart and select **Export Image** from the resulting local menu. This brings up the save file dialog. Choose jpeg as the file type, type the file name, and press save. Specify the compression/resolution in the Jpeg Options Dialog. Click **Cancel** to terminate the process, or click **Save** to save the jpeg.

---

## Saving Admittance Matrix and Jacobian Information

---

The **Save Ybus or Power Flow Jacobian Dialog** is used to store the power system bus admittance matrix (Ybus) and/or the power flow Jacobian in a text format that can be easily read into other programs such as MATLAB®. This dialog is primarily designed for users doing power system analysis research. The dialog has the following fields.

### Ybus in MATLAB Format

#### Filename for Saving Ybus

Enter the name of the file in which to store the Ybus data. The Ybus data is stored using the MATLAB sparse matrix format in the matrix **Ybus**. If the **Include Bus Voltages** field is checked, then the bus voltages are also stored, but in the vector **V**.

#### Save Ybus in MATLAB Format

Click this button to save the Ybus.

### Power Flow Jacobian in MATLAB Format

#### Jacobian Save File

Enter the name of the file in which to store Jacobian data. The Jacobian is stored using the MATLAB sparse matrix format in the matrix **Jac**.

#### Jacobian ID Save File

Enter the filename to store the text identifier information. This information is used to translate the bus numbering convention used in the Jacobian and Ybus files with the actual bus number and name in the case.

#### File Type

Choose the type of MATLAB file you wish to save as. The **MATLAB .M Format** is the more common text format used for directly loading MATLAB information. The **Text for MATLAB Ascii** is for use with MATLAB's ability to read Ascii files. The Ascii file type can be read into MATLAB much faster than the traditional .M files.

#### Jacobian Form

Select **Rectangular** to store the rectangular form of the Jacobian, or **Polar** to store the polar form of the Jacobian.

#### Save Jacobian

Click this button to save the Jacobian and object identifier information.

---

## Chapter 4: Building a Oneline Diagram

---

The oneline objects provide a key means of interacting with the power flow simulation. This chapter provides a reference on how to insert each of the different types of objects shown on the oneline diagram, and an overview of Edit Mode. Note that when you are using the Simulator, you can obtain context sensitive help for oneline objects by positioning the cursor on the desired object and pressing the F1 key.

The following material is included:

- Edit Mode Overview
- Areas, Zones and Owners
- Buses
- Substations
- Generators
- Loads
- Transmission Lines
- Transformers
- Series Capacitors
- Switched Shunts
- Interfaces
- Oneline Links
- Background Objects and Text

---

## Edit Mode Overview

---

The Edit Mode is used to create and/or modify cases and onelines. You can use the Edit Mode to create a case from scratch or to modify existing power flow cases stored in PowerWorld PWB or AUX files, in the PTI Raw data format, GE PSLF EPC data format, or the IEEE Common Format. New users may wish to view step-by-step tutorials on either

[Creating a New Case from Scratch](#)

[Creating Onelines for an Existing Power Flow Case.](#)

To enter the Edit Mode, select the **Edit Mode** button in the **Mode** ribbon group on any of the ribbon tabs.

A powerful capability of the Simulator is its ability to a case by graphically placing/editing on a oneline diagram. These display objects consist of both power system devices, such as buses, generators, and transmission lines; and that show various system parameters, provide descriptive text, or function as a static background.

Simulator's oneline diagrams illustrate the current state of the components of the power system. Most display objects correspond to records in the underlying power system model, but not all records in the power system model need to have an associated display object. In fact, for large system models, it may be that most of the system will not be illustrated. In such cases, you will want to devote more detail to the more critical areas of the system so as not to clutter the view. Furthermore, it is possible to associate more than one oneline with a single power flow case, and a single oneline may be associated with multiple cases. This great flexibility can prove to be a big time-saver. Please see [Relationship Between Display Objects and the Power System Model](#) for a more thorough discussion.

---

## Edit Mode General Procedures

---

In order to simplify the process of graphically constructing a power flow case, Simulator's drawing interface obeys the following conventions for most objects:

### Inserting a New Object

- Go to the [Draw](#) ribbon tab and use the menus on the **Error! Hyperlink reference not valid.** ribbon group to add a single oneline object type you would like to add to the oneline.
- Left-click on the location on the display where you would like to position the object.
- Once the object is placed, Simulator displays a dialog box that allows you to specify various options for the object.
- If desired, use all the options available on the [Formatting](#) ribbon group on the [Draw](#) ribbon tab to change the appearance of the object once it has been placed.

### Moving, Resizing, and Rotating an Existing Object

- Select the object by clicking on the object with the left mouse button. Handles are displayed around the object to indicate it has been selected.
- To move the position of the object, place the mouse anywhere on the object except at a handle location. Then drag the object around the screen by holding the left mouse button down.
- To change the size of an object using the mouse, first select the object. Then place the mouse on one of the object's resizing (white) handles. The cursor will change to either a horizontal, vertical or diagonal two-headed arrow shape. Then drag the mouse to change the object's size. You can also specify the size of most objects using their dialog boxes.
- To rotate an object, first select the object. Then place the mouse over the rotation (green) handle in the upper left corner of the selected object. The cursor will change to a rotation symbol. Clicking and dragging will then rotate the object. Note that if an object (e.g., a pie chart or transmission line) does not have a rotation handle in the upper left, the object cannot be rotated.

### Viewing/Modifying Object Parameters

- To view and/or change the options associated with a single object, right-click on the object. This either displays the object's dialog box directly, or it displays the object, from which you can elect to see the object's dialog box.

### Selecting Several Objects to Modify Their Appearance

Hold down the Shift key while clicking objects on the screen to select several objects at once. You may then change the objects' attributes, such as , by using all the options available on the [Formatting](#) ribbon group on the [Draw](#) ribbon tab.

To move the objects that are selected, click and hold down the left mouse button on any of the selected objects, drag the selected objects to a new location, and release the mouse button to place them. You can also move the selected objects by holding the SHIFT key and simultaneously using the UP, DOWN, LEFT, or RIGHT arrow keys (or use the Home, End, Page Up and Page Down keys to move objects).

To select a set of objects use the many options available on the [Select](#) ribbon group on the [Draw](#) ribbon tab to select a group of objects using a [rectangle, ellipse or polygon selector](#). You can also select only objects that meet some given criteria by using [Select by Criteria](#). As an example, you can use Select By Criteria to select all the 345 kV transmission lines in a case. See [Select](#) ribbon group for more detailed help.

### Changing An Object's Screen Appearance

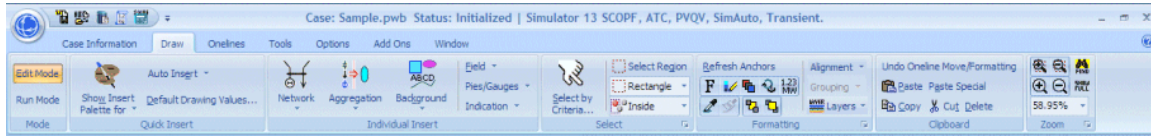
Use all the options available on the [Formatting](#) ribbon group on the [Draw](#) ribbon tab to change the screen appearance of either a selection of objects or the entire display.

---

## Draw Ribbon Tab

---

The Draw ribbon tab is primarily used to draw new oneline diagrams or edit existing onelines by adding, moving, formatting, or resizing existing oneline objects. Most of the options on the Draw ribbon tab are only available in [Edit Mode](#). The Draw ribbon tab is shown below.



The Draw ribbon consists of seven ribbon groups described as follows

### Mode Ribbon Group

#### Edit Mode

Switches the program to [Edit Mode](#), which can be used to build a new case or to modify an existing one.

#### Run Mode

Switches the program to [Run Mode](#), which can be used to perform a single Power Flow Solution or a timed simulation with animation.

### Quick Insert Ribbon Group

The Quick Insert ribbon group contains buttons for creating a oneline diagram for a system model you have already read into Simulator. If you are building out your network (or even substation, area, or zone diagram), then using the Insert Palettes along with the Auto Insert routines is most efficient. For detailed help see the [Quick Insert](#) ribbon group. If you would instead like help on inserting a single object on your oneline diagram see the [Individual Insert](#) ribbon group topic.

### Individual Insert Ribbon Group

The Individual Insert ribbon group contains menus which provide access to buttons for inserting individual oneline objects to the oneline diagram. If you are building out your network (or even substation, area, or zone diagram), then it is much more efficient to use the [Quick Insert](#) ribbon group. However, for complete access to all objects including those that can not be "quick - inserted", see the help under the [Individual Insert](#) ribbon group topic.

### Select Ribbon Group

The Select ribbon group contains options to help select multiple oneline objects simultaneously. For more detailed help see the [Selection](#) ribbon group topic for more detailed help.

### Formatting Ribbon Group

The Format ribbon group contains options for formatting the selected objects on your active oneline. For more detailed help see the [Formatting](#) ribbon group topic for more detailed help.

### Clipboard Ribbon Group

The Clipboard ribbon group contains options for copying and pasting oneline display objects. For more detailed help see the [Clipboard](#) ribbon group topic for more detailed help.



### **Zoom Ribbon Group**

The Zoom ribbon group contains buttons for navigating a online diagram. For more detailed help see the [Zoom](#) ribbon group topic

---

## Online Display Options

---

The [Online Display Options Dialog](#) allows you to customize the appearance of the presently selected oneline diagram. To view this dialog, either select **Online Display Options** from the **Online Options** ribbon group on the [Options](#) ribbon tab, or choose **Online Display Options** from the oneline's [local menu](#). Please see the [Online Display Options Dialog](#) help for more information.

---

## Anchored Objects

---

While in [Edit Mode](#), Simulator allows certain objects to be attached, or *anchored*, to another object, called the *anchor*. When an object that functions as an anchor is moved, all objects that are anchored to it will move with it. This feature can be very useful when you move objects around the oneline diagram in Edit Mode.

Anchoring has the property of "stacking" in Simulator. In other words, one object is anchored to another, which is in turn anchored to yet another. The best way to describe this is by example. A generator text field can be anchored to a generator. The generator, in turn, can be anchored to its terminal bus. If you move the terminal bus, both the generator and its anchored fields also move with the bus. However, if you just move the generator itself, only the generator fields will move with it. The bus and all other objects anchored directly to the bus remain in their original location.

If anchors are deleted it is generally necessary to reset anchors on anchored objects go to the [Draw](#) ribbon tab and choose **Refresh Anchors** on the [Formatting](#) ribbon group.

There are several types of anchored objects:

### [Buses](#)

[Loads](#), [generators](#), [switched shunts](#), [bus fields](#), [interfaces](#), [transformers](#), [transmission lines](#) and [voltage gauges](#) may be anchored to their associated bus. When the anchor bus is moved, these anchored objects will move with it.

### [Substations](#)

Similar to buses, any [loads](#), [generators](#), [switched shunts](#), [bus fields](#), [interfaces](#), [transformers](#), and [transmission lines](#) may be anchored to their associated substations. When the anchor substation is moved, these anchored objects will move with it.

### [Generators](#)

[Generator fields](#) may be anchored to their associated generator. When the anchor generator is moved, these anchored fields will move with it.

### [Loads](#)

[Load fields](#) may be anchored to their associated load. When the anchor load is moved, these anchored fields will move with it.

### [Switched Shunts](#)

[Switched shunt fields](#) may be anchored to their associated switched shunt. When the anchor switched shunt is moved, these anchored fields will move with it.

### [Area/Zone/Super Area Objects](#)

[Interfaces](#) can be anchored to area/zone/super area objects.

### [Lines and Transformers](#)

[Circuit breakers](#), [line flow pie charts](#), and [line fields](#) may be anchored to their associated line/transformer. When the line/transformer is moved, these anchored objects will move with it.

### [Interfaces](#)

[Interface fields](#) and [interface pie charts](#) can be anchored to their associated interface.

---

## Inserting and Placing Multiple Display Fields

---

Simulator provides a convenient method of adding display fields to a variety of oneline display objects and placing them in default positions relative to the display object. Unlike the field placement implemented by selecting **Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab, the method described here allows you to place multiple fields around a display object in a single operation. This option is available for several different types of objects, including buses, generators, lines and transformers, shunts, and loads. The option can always be accessed by right clicking the display object of interest and selecting the **Add New Fields Around ...** option from the resulting local menu. This brings up the **Insert New Fields around Selected Objects Dialog**. This dialog is divided into several tabs by object type. Only tabs that correspond to the type of objects you have selected will be available.

The tabs for inserting and placing multiple display fields for the various display objects (buses, generators, lines/transformers, shunts, and loads) are virtually identical in content. The tab illustrates the possible locations of the display fields for the various orientations of the display object. For example, since buses may be oriented either horizontally or vertically, the dialog shows how each of the eight possible bus fields would be positioned for each of the two orientations. Generators, loads, and shunts each have four possible orientations, so the dialog identifies the locations for the possible fields for each of the four orientations. Transmission lines and transformers can assume only one orientation, and the dialog will thus show the possible field locations for that single orientation.

Each field location is identified on the illustrations with a label of the form Pos #. In order to modify the settings, move your mouse over the position on the dialog you want to change and click. This will bring up the appropriate Field Options Dialog such as the [Bus Field Options](#), [Generator Field Options](#), [Load Field Options](#), [Switched Shunt Field Options](#), or the [Line Field Options](#). Simply select the field you want and choose **OK**. If you would like to set a default field to "none", click **Remove Field** instead of **OK**.

Click **OK** to implement your choices for field additions and placement, or click **Cancel** to discard the changes.

This help topic has gone over how to add new fields to existing display objects. It should be noted from this discussion that all objects have an associated set of default fields that will be added to the oneline when the objects are originally inserted. You may redefine the default fields selecting **Default Drawing Values** from the [Quick Insert](#) ribbon group on the [Draw](#) ribbon tab and modifying the [Default Drawing Options Dialog](#).

---

## Area Display Objects

---

Area records in the load flow data can be displayed as objects on a Simulator oneline diagram. This can be useful for building a diagram on which you also want to include representations of groups of devices by area.

To insert an Area object, select **Aggregation > Area** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then left-click on a oneline diagram in the location where the Area object should be placed. This will open the [Area Display Options](#) dialog, which will allow you to choose the area information to display, and set parameters for the displayed object. It is important to note that area records, unlike other objects like buses and transmission lines, cannot be added to the load flow data graphically. To add a new area record to the load flow case, you need to change the area designation of a device in the load flow case to a new area number, or you need to insert a new area from the [Area Records](#).

Area objects can also be added by using the [Area insert palette](#).

---

## Area Display Options Dialog

---

When you insert an area display object on a oneline diagram, Simulator opens the Area Display Options dialog. This dialog is used to control various display and identity attributes of the area display object. The dialog contains the following fields:

### Number

This dropdown box lists the number of all areas in the case. Use this control to associate the display object with the correct area.

### Name

If you would prefer to search through areas by name rather than by number, use the *Name* dropdown box to see a list of names of all the areas in the case.

### Show Record Type Prefix

Check this box if you wish to place the prefix **Area** before the name/number caption in the object.

### Prefix Text

Specify additional prefix text to be added before the Record Type prefix.

The remainder of the choices presented on the *Area Display Options* dialog pertain to the object's display appearance.

### Style

Choose whether the display object should appear as a rectangle, rounded rectangle, or an ellipse.

### Caption

Indicate how the display object should be identified to the user: by name, number, or both.

### Width, Height

The dimensions of the new display object.

Click **OK** to save your selections and add the object to the oneline, or choose **Cancel** to terminate the addition.

---

## Zone Display Objects

---

Zone records in the load flow data can be displayed as objects on a Simulator oneline diagram. This can be useful for building a diagram on which you also want to include representations of groups of devices by zone.

To insert a Zone object, select **Aggregation > Zone** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then left-click on a oneline diagram in the location where the Zone object should be placed. This will open the [Zone Display Options](#) dialog, which will allow you to choose the zone information to display, and set parameters for the displayed object. It is important to note that zone records, unlike other objects like buses and transmission lines, cannot be added to the load flow data graphically. To add a new zone record to the load flow case, you need to change the zone designation of a device in the load flow case to a new zone number, or you need to insert a new zone from the [Zone Records](#).

---

## Zone Display Options Dialog

---

When you insert a zone display object on a oneline diagram, Simulator opens the Zone Display Options dialog. This dialog is used to control various display and identity attributes of the zone display object. The dialog contains the following fields:

### Number

This drop down box lists the number of all zones in the case. Use this control to associate the display object with the correct zone.

### Name

If you would prefer to search through zones by name rather than by number, use the *Name* drop down box to see a list of names of all the zones in the case.

### Show Record Type Prefix

Check this box if you wish to place the prefix **Zone** before the name/number caption in the object.

### Prefix Text

Specify additional prefix text to be added before the Record Type prefix.

The remainder of the choices presented on the *Zone Display Options* dialog pertain to the object's display appearance.

### Style

Choose whether the display object should appear as a rectangle, rounded rectangle, or as an ellipse.

### Caption

Indicate how the display object should be identified to the user: by name, number, or both.

### Width, Height

The dimensions of the new display object.

Click **OK** to save your selections and add the object to the oneline, or choose **Cancel** to terminate the addition.



---

## Super Area Display Objects

---

Super area records in the load flow data can be displayed as objects on a Simulator online diagram. This can be useful for building a diagram on which you also want to include representations of groups of devices by super areas.

To insert a Super Area object, select **Aggregation > Super Area** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then left-click on a online diagram in the location where the Super Area object should be placed. This will open the [Super Area Display Options](#) dialog, which will allow you to choose the super area information to display, and set parameters for the displayed object. It is important to note that super area records, unlike other objects like buses and transmission lines, cannot be added to the load flow data graphically. To add a new super area record to the load flow case, you need to insert a new super area from the [Super Area Records](#).

---

## Super Area Display Options Dialog

---

When you insert a super area display object on a oneline diagram, Simulator opens the Super Area Display Options dialog. This dialog is used to control various display and identity attributes of the super area display object. The dialog contains the following fields:

### Number

This field is unused for super area objects. Super areas are identified by a unique super area name instead.

### Name

Use the *Name* dropdown box to see a list of names of all the super areas in the case.

### Show Record Type Prefix

Check this box if you wish to place the prefix **Super Area** before the name/number caption in the object.

### Prefix Text

Specify additional prefix text to be added before the Record Type prefix.

The remainder of the choices presented on the *Super Area Display Options* dialog pertain to the object's display appearance.

### Style

Choose whether the display object should appear as a rectangle, a rounded rectangle, or an ellipse.

### Caption

Indicate how the display object should be identified to the user: by name, number, or both.

### Width, Height

The dimensions of the new display object.

Click **OK** to save your selections and add the object to the oneline, or choose **Cancel** to terminate the addition.

---

## Owner Display Objects

---

Owner records in the load flow data can be displayed as objects on a Simulator online diagram. This can be useful for building a diagram on which you also want to include summary objects by owner, in which information about the generation and load of the owner are indicated with the object.

To insert an Owner object, select **Aggregation > Owner** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then left-click on a online diagram in the location where the owner object should be placed. This will open the [Owner Display Options](#) dialog, which will allow you to choose the owner information to display, and set parameters for the displayed object.

---

## Owner Display Options Dialog

---

When you insert an owner display object on a oneline diagram, Simulator opens the Owner Display Options dialog. This dialog is used to control various display and identity attributes of the owner display object. The dialog contains the following fields:

### Number

The owner number.

### Name

The owner name.

### Show Record Type Prefix

Check this box if you wish to place the prefix **Owner** before the name/number caption in the object.

### Prefix Text

Specify additional prefix text to be added before the **Owner** prefix.

The remainder of the choices presented on the *Owner Display Options* dialog pertain to the object's display appearance.

### Style

Choose whether the display object should appear as a rectangle or as an ellipse.

### Caption

Indicate how the display object should be identified to the user: by name, number, or both.

### Width, Height

The dimensions of the new display object.

Click **OK** to save your selections and add the object to the oneline, or choose **Cancel** to terminate the addition.

---

## Area Fields on Onelines

---

Area fields are used to show various values associated with a particular area of the power system. Right clicking on the area field displays the [Area Field Dialog](#).

### Edit Mode

To enter a new area field, first select **Field > Area Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then, click on or near the area object or a bus in the area for which you want to add a field. This calls up the [Area Field Dialog](#). Verify that the area number is correct. By default, this value is the number of the area associated with the closest area object, or if there are no area objects, the closest bus. Enter the total number of digits the field should display as well as the number of digits to the right of the decimal point. Depending on what the field is designed to display, you may need to enter an additional area number. Finally, select the field type. Click **OK** to save the field or **Cancel** to abort the operation.

With most types of area fields, an **Area Number** of 0 is valid and defines the field as showing values for the entire system.

To modify the parameters of an existing area field, position the cursor anywhere on the area field and right-click. This again brings up the [Area Field Dialog](#). Use the [Formatting](#) options to change various display attributes for the area field, including its font and background color.

---

## Zone Fields on Onelines

---

Zone field objects are used to show different values associated with zones and the system. This dialog is used to view (and in a few cases modify) the parameters associated with these zone fields. To reach this dialog, select **Field > Zone Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab and click the desired field location on a oneline diagram in Edit Mode, or right-click on an existing zone field on a oneline diagram.

### Zone Number

Zone number associated with the field. When you insert fields graphically, this field is automatically set to the zone number associated with the closest bus on the oneline. With most types of zone fields, a *Zone Number* of 0 is valid and defines the field as showing values for the entire system.

### Find...

If you do not know the exact zone number you are looking for, you can click this button to open the [advanced search engine](#).

### Total Digits in Fields

Total number of digits to show in numerical fields.

### Digits to Right of Decimal

Number of digits to show to the right of the decimal point in numerical fields.

### Rotation Angle in Degrees

The angle at which the text is to appear on the oneline diagram.

### Other Zone Number

Some of the fields, such as **MW Flow to Other Zone**, require that a second area be specified. If applicable, enter the second (other) zone here.

### Delta per Mouse Click

This value is used only with the **Sched Flow to Other Zone** field type. When there is a nonzero entry in this field, and the field type is **Sched Flow to Other Zone**, a spin button is shown to the right of the zone field. When the up spin button is clicked, the flow to the other zone is increased by this number of MW; when the down button is clicked, the scheduled flow is decreased by this amount.

### Field Value

Shows the current output for the zone field. Whenever you change the **Type of Field** selection, this field is updated.

For the **Sched Flow to Other Zone** field type only, you can specify a new value in MW. Exports are assumed to be positive.

### Field Prefix

An optional string that precedes the field value.

### Anchored

If the field is associated with a Zone Object on the diagram, the field can be anchored to the object so that if the object gets moved on the diagram, the field will move with it.

### Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

### Type of Field

Used to determine the type of zone field to show.

#### Name

Zone name.

#### Number

Zone number (1 - 999).

#### MW Load, Mvar Load

If the zone number is nonzero, then these fields show Total MW or Mvar load for the zone. If the zone number is zero, these fields show the total load in the entire system.

#### MW Generation, Mvar Generation

If the *Zone Number* is nonzero, then these fields show Total MW or Mvar generation for the zone. If the *Zone Number* is zero, these fields show the total generation in the entire system.

#### MW Shunts, MVR Shunts

If the *Zone Number* is nonzero, then these fields show Total MW or Mvar shunt compensation for the zone. If the *Zone Number* is zero, these fields show the total shunt compensation in the entire system.

#### MW Flow to Other Zone, Mvar Flow to Other Zone

Total MW or Mvar flow from the zone specified in the *Zone Number* field to the zone specified in the *Other Zone Number* field. The Zone Number field must correspond to a valid zone. If the *Other Zone Number* field is zero, this field shows the zone's total MW or Mvar exports.

#### MW Losses, MVAR Losses

If the *Zone Number* is nonzero, then these fields show Total MW or Mvar losses for the zone. If the *Zone Number* is zero, these fields show the total real or reactive losses in the entire system.

#### Load Schedule Multiplier

Indicates the current value of the MW multiplier applied to the zone's loads.

#### Select a Field

Allows you to choose any other field associated with a zone not explicitly listed as an option in the Type of Field selection.

Select *OK* to save changes and close the dialog or *Cancel* to close dialog without saving your changes.

---

## Super Area Fields on Onelines

---

To display certain information about a super area, such as MW Load or MVAR losses, insert a super area field. This can be done in Edit Mode by selecting **Field > Super Area Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. This will bring up the [Super Area Field Information](#). From here you can choose which super area to describe, how many digits in the field, and how many digits to the right of the decimal. There are also 12 different field options to choose from. If a field value is not defined, question marks will be shown.



---

## Owner Fields on Onelines

---

Owner field objects are used to show different values associated with owners. This dialog is used to view and modify the parameters associated with these fields. To insert an Owner field, click on **Field > Owner Field** in the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.

### Owner Number

Owner number associated with the field. When you insert fields graphically, this field is automatically set to the owner number associated with the closest bus on the oneline.

### Find by Number

To switch to a different owner in the field options dialog, you can enter the number in the **Owner Number** field, and press the **Find by Number** button to update the dialog with information for the new owner.

### Owner Name

Name of the owner whose information is presently being displayed in the dialog.

### Find by Name

To switch to a different owner in the field options dialog, you can enter the name in the **Owner Name** field, and press the **Find by Name** button to update the dialog with information for the new owner.

### Total Digits in Fields

Total number of digits to show in the field.

### Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

### Field Prefix

An optional string that precedes the field value.

### Rotation Angle in Degrees

The angle at which the text is to appear on the oneline diagram.

### Field Value

Shows the current output for the owner field. Whenever you change the **Type of Field** selection, this field is updated.

### Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

### Type of Field

Used to determine the type of owner field to show. You can choose from one of the six default fields, or click on the **Find Field...** button to choose from a list of all available owner fields.

---

## Bus Display Objects

---

In power system analysis, the term "bus" is used to refer to the point where a number of electrical devices, such as lines, loads or generators, join together. On the oneline diagram, buses are usually represented with either a thick horizontal line or a thick vertical line. The bus thickness and color can be customized.

Right-clicking on the bus will display its local menu. The local menu offers you the chance to view the corresponding [Bus Dialog](#), the [Quick Power Flow List](#), and the [Bus View Display](#). When the application is in Edit Mode, the local menu will also allow you to add bus fields to the bus and to insert any undrawn buses connected to the selected bus. [Bus Fields](#) are often placed close to the bus to indicate its voltage magnitude, voltage angle, and other relevant information.

### Edit Mode

To add a new bus to the case graphically, follow this simple procedure:

- Select **Network > Bus** from the [Individual Insert](#) ribbon group of the [Draw](#) ribbon tab. This prepares Simulator to insert a new bus.
- Left-click on the oneline background at the location where you want to place the new bus. This invokes the [Bus Option Dialog](#).
- Use the Bus Option Dialog Box to specify the number, name, size, thickness, orientation, area, zone, and nominal voltage of the bus, as well as the load and shunt compensation connected to it. Every bus must have a unique number.
- Click **OK** on the Bus Option Dialog to finish creating the bus and to close the dialog. If you do not wish to add the bus to the case, click **Cancel**.

If you are simply adding a symbol to the oneline diagram for a bus that has already been defined in the case, many of the parameters you are asked to specify in step three will be filled in for you.

To modify the parameters for an existing bus, position the cursor on the bus and right-click to invoke the bus' local menu. From the local menu, choose **Bus Information Dialog** to view the associated [Bus Dialog](#). You may change any of the parameters specified there. When a bus' number is changed, the bus numbers associated with all of the devices attached to that bus are also automatically changed. To renumber several buses simultaneously, please see [Bus Renumbering Dialog](#).

To modify any aspect of a bus' appearance, first select the bus, and then click any of the format buttons found in the [Formatting](#) ribbon group on the [Draw](#) ribbon tab. You can change the length of the bus (but not its thickness) by dragging the bus' resizing handles.

In order to delete an existing bus after selecting it, use either the **Cut** button, found in the [Clipboard](#) ribbon group on the [Draw](#) ribbon tab, to preserve a copy of the bus on the Windows clipboard, or **Delete** button, found in the [Clipboard](#) ribbon group on the [Draw](#) ribbon tab, to remove the bus without copying it to the clipboard. You will be asked whether you want to remove both the display object and its associated bus record, or merely the display object, leaving the bus in the power flow model. If you will never be deleting a record from the power system model, you may also choose the option labeled **Always Delete Objects Only**. Be careful when deleting existing buses with attached devices. An error will occur during validation if you do not also delete the attached devices or attach them to other buses.

---

## Bus Fields on Onelines

---

Bus field objects are used primarily to indicate various quantities associated with bus devices. Furthermore, some bus field types, which are distinguished by an integrated spin button, may be used to easily change bus device properties.

### Run Mode

Right clicking on a bus field gives you the option to open the [Bus Field Dialog](#) or the [Bus Information Dialog](#).

### Edit Mode

Simulator offers two options for adding bus fields to a oneline in Edit Mode. If you need to enter only a single field, the easier approach may be to choose **Field > Bus Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab and then select the bus to which you want to add the field. This invokes the [Bus Field Dialog](#). Enter the bus number associated with the device (the default is the closest bus to the field), the total number of digits to show, and the number of digits to the right of the decimal point. An optional Field Prefix can be used. Choosing to Include Suffix will include relevant units for the type of field selected. The field can be Anchored to the bus that it represents and the field can be rotated by a specified Rotation Angle in Degree. Finally, select the type of field to show.

The second approach for adding new bus fields entails right-clicking the bus and selecting **Add New Fields Around Bus** from the resulting local menu. See [Inserting and Placing Multiple Display Fields](#) for more details.

To modify the parameters of an existing bus field, position the cursor anywhere on the object and right-click. This brings up the [Bus Field Dialog](#). Choose any of the format buttons on the [Formatting](#) ribbon group on the [Draw](#) ribbon tab to change various display attributes of the field, including its font and background color.

---

## Old Voltage Gauges

---

Voltage gauges provide a way to visualize the voltage of a bus relative to its high- and low-voltage alarm limits. A gauge looks very much like a thermometer. As the temperature changes, the height of the mercury in the thermometer moves up and down. One reads the temperature measured by the thermometer by noting the marking that matches the top of the mercury. Voltage gauges in Simulator work the same way. A voltage gauge has three markings on its side, each of which identifies a key per-unit voltage level. Specifically, these three markings locate the minimum, maximum, and target per-unit voltages. Often, the target voltage level is the nominal voltage, but this is not a requirement. Inside the gauge is a filled region. The default color of the filled region is blue, but this can be changed. When you create the voltage gauge, you associate it with a bus, and you specify its fill color and its minimum, maximum, and target voltage levels. Once the gauge has been placed on a display, it will reveal changes in its associated bus's voltage by varying the height of its filled region. This tool was introduced to provide an alternative to voltage contours to show the variation of voltage across a region. [Contouring](#) can reveal the variation of only a single quantity at a time. For example, it is impossible to contour bus voltage magnitude and bus voltage phase angle simultaneously. Voltage gauges are helpful because they allow you to show the voltage profile superimposed on a contour of some other quantity.

To add a voltage gauge to a display, switch to Edit Mode and select **Pies / Gauges > Old Gauges > Bus** from the **Individual Insert Ribbon Group** on the [Draw](#) ribbon tab. The cursor will become a crosshair. Click on the oneline diagram where you would like the new voltage gauge to appear. The [Voltage Gauge Options Dialog](#) will appear. Use this dialog to define the minimum, maximum and target voltages for the voltage gauge, as well as its fill color and whether it should be anchored to its associated bus. After you click the "OK" button, the Voltage Gauge Options Dialog will close, and the new voltage gauge will appear.

Once a gauge has been placed on the oneline, the height of its filled region will change as the voltage of its associated bus changes. To modify any of the characteristics of the gauge, such as its key voltage levels, fill color, and anchor setting, simply right-click on the bus voltage gauge to open the Voltage Gauge Options Dialog again.

---

## Old Voltage Gauge Options Dialog

---

The Voltage Gauge Options Dialog is used to define and configure a [bus voltage gauge](#). A bus voltage gauge is associated with a particular bus and reveals the bus's voltage relative to specified minimum, maximum, and target per-unit voltage levels. The height of the colored column in the voltage gauge indicates the bus's voltage relative to these markings.

The dialog has the following controls:

### Number, Name, and Find

Use the **Number** and **Name** dropdown boxes to identify the bus to which you want the gauge to correspond. Select a bus number from the Number dropdown box to identify the bus by number, and a bus name from the Name dropdown box to identify a bus by name. It may be more convenient to press the **Find** to open the [Find Dialog](#), which allows you to specify a bus by either name or number using wildcards. When you first open the Bus Voltage Gauge Dialog, the bus name and number will correspond to the bus object that was closest to the point where you clicked.

### Minimum, Target, and Maximum

Use these three spin edit boxes in the **Voltage levels** group box to specify the minimum, maximum, and target per-unit voltage levels. These settings determine where on the gauge its three markings will be drawn.

The Minimum and Maximum value will be taken from the Limit Monitoring settings for the current bus, unless you uncheck the option "*Set Limits According to Current Limit Monitoring Settings*".

### Fill color

The Fill color box reveals the color that will be used to paint the filled region of the gauge. Click on the Fill color box to open a Color Dialog, which you may then use to specify a different color.

### Anchored

A bus voltage gauge is said to be [anchored](#) if, when you move its associated bus object, it moves with it. Check the Anchored check box to ensure that the gauge will move with its associated bus. Otherwise, when you move its associated bus object, the gauge will stay in its current position.

### OK, Help, and Cancel

Click OK to finalize your settings. This will create a new voltage gauge object if you are trying to create one from scratch, or it will modify the appearance and settings of an existing one if you have chosen to modify one that has already been defined. Click Cancel to dispose of your changes. Click the Help button to reveal this help text.

---

## Substation Display Objects

---

Substations in Simulator define a group of buses that are closely connected. Each bus can belong to either one substation or no substation (called unassigned). By default in Simulator, all buses are not assigned to a substation. This is done because all traditional text file power flow formats do not include information regarding a bus' substation.

Substations are represented on the oneline as a rectangle with the name of the substation inside it. Other information about the substation is also displayed on the rectangle:

- The upper left corner displays a generator symbol if generation exists in the substation.
- The upper right corner displays a load symbol if load exists in the substation.
- The lower right corner displays a shunt symbol if shunts exist in the substation.
- The lower left corner displays the number of buses inside the substation.
- The lower middle displays the maximum voltage level in the substation.

You can also customize the size, colors, and font name and style of a substation object. Note however that the font size of the substation object is automatically changed by Simulator as you change the size of the rectangle.

Right-clicking on the substation will display its local menu. The local menu offers you the chance to view the corresponding [Substation Information Dialog](#) and the [Substation View Display](#).

---

## Substation Fields on Onelines

---

To display certain information about a substation, such as MW Load or MVAR losses, insert a substation field. This can be done in Edit Mode by selecting **Fields > Substation Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. This will bring up the [Substation Field Options](#) dialog. From here you can choose which substation to describe, how many digits in the field, and how many digits to the right of the decimal. There are also 8 different field options to choose from. If a field value is not defined, question marks will be shown.

---

## Generator Display Objects

---

Generators are represented on the oneline as circles with a "dog bone" rotor inside. Multiple generators at a bus are allowed, with each being distinguished by a unique character identifier.

Each generator symbol (except that corresponding to the slack) is equipped with a circuit breaker that can be used to change the status of the generator. You may toggle the generator status by clicking on the circuit breaker while in Run Mode.

[Generator fields](#) are often placed close to the generator on the oneline to indicate the generator's MW/Mvar output.

### Run Mode

When [animation](#) is active, the default flow of the arrows emerging from the generator is proportional to its MW output. You can customize the appearance of this flow using the Animated Flows Tab of the [Online Display Options Dialog](#).

Right clicking on the generator brings up the generator submenu. This menu is used to display a variety of information about the generator, including

- [Generator Information Dialog](#)
- [Input-output curve](#)
- [Fuel cost curve](#)
- [Incremental cost curve](#)
- [Heat-rate curve](#)
- [Reactive capability curve](#)

### Edit Mode

To add a new generator to the case, select **Network > Generator** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then, place the cursor on the bus where you would like to attach the generator, and click with the left mouse button. This calls up the [Generator Dialog](#). The bus number is automatically determined from the bus to which you attached the generator. The ID field contains an alphanumeric ID used to distinguish multiple generators at a bus. The default is '1'.

Enter the size, the thickness of the lines (in pixels) used to display the device, orientation, and other parameters for the generator. Each generator automatically contains a switch for connecting or disconnecting the device in Simulator. Select OK to add the generator. If you do not want to add the generator to the case, select Cancel.

To modify the parameters for an existing generator, position the cursor on the generator and right-click. This again brings up the Generator Dialog. You can then change any parameter (be careful in renumbering an existing generator). Use the [Formatting](#) ribbon group commands to change the color, line thickness, and other display parameters.



---

## Generator Fields on Onelines

---

Generator field objects are used primarily to indicate various quantities associated with generation devices. Furthermore, some generator field types, which are distinguished by an integrated spin button, may be used to change generation device properties.

### Run Mode



For generator fields with an associated spin button, clicking on the up/down arrows will change the value of the associated field.

Right clicking on a generator field gives you the option to open the [Generator Field Dialog](#) or the [Generator Information Dialog](#).

### Edit Mode

Simulator offers two options for adding generator fields to a oneline in Edit Mode. If you need to enter only a single field, the easier approach may be to choose **Field > Generator Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab and click near generator for which you want to add the field. This invokes the [Generator Field Dialog](#). Enter the bus number associated with the device (the default is the bus associated to the closest generator to the field), the ID field, the total number of digits to show, and the number of digits to the right of the decimal point. Next, select the type of field to show. For generator actual MW and Setpoint MW types and load MW and Mvar fields, specify a nonzero value in the *Delta per Mouse Click* to design a bus field with an integrated spin control. The Gen AGC Status field is used to display the automatic generation control status of the generator. The user can toggle this status in Simulator by clicking on the field. Likewise, the Gen AVR Status field is used to display the automatic voltage regulation status of the generator. Again, the user can toggle this status by clicking on the field.

The second approach for adding new generator fields entails right-clicking the bus and selecting *Add New Fields Around Generator* from the resulting local menu. Please see [Inserting and Placing Multiple Display Fields](#) for more details.

To modify the parameters of an existing generator field, position the cursor anywhere on the object and right-click. This brings up the [Generator Field Dialog](#). Choose , or  from the [Formatting](#) ribbon group to change various display attributes of the field, including its font and background color.

---

## Load Display Objects

---

Simulator models aggregate load at each system bus. Multiple loads at a bus are allowed. Each load object on the oneline comes equipped with a circuit breaker. The status of the load corresponds to the status of its circuit breaker. A circuit breaker is closed if it appears as a filled red square, and it is open if it appears as a green square outline. In Run Mode, you may toggle the status of the load by clicking on its associated circuit breaker.

[Load fields](#) are often placed close to the loads on the oneline to indicate their MW/Mvar value.

### Run Mode

When [animation](#) is active, the flow of the arrows into the load is proportional to its current MW load. You can customize the appearance of this flow using the Animated Flows Tab of the [Oneline Display Options Dialog](#).


Right clicking on a load (bus) field gives you the option to open the [Load Field Dialog](#) or the [Load Dialog](#).

### Edit Mode

To add a new load to the case, select **Network > Load** from [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then, select the bus to which you want to attach the load with the left mouse button. This calls up the [Load Dialog](#). The bus number is automatically determined from the bus to which you attached the load. The ID field contains a two-character ID used to distinguish multiple loads at a bus. The default ID is 1.

Enter the size, the thickness of the lines [in pixels] used to display the device, the orientation, and the base MW and Mvar load values for the device. Usually, only the Constant Power fields are specified as nonzero. The Constant Current and Constant Impedance fields are used to specify loads that vary with voltage. Constant current loads vary proportionally with bus voltage, while constant impedance loads vary with the square of the voltage. Specify the constant current and constant impedance values assuming one per-unit voltage.

Select OK to add the load. If you do not want to add the load to the case, select Cancel.

To modify the parameters for an existing load, position the cursor on the load and right-click. Select *Load Information Dialog* from the local menu to invoke the [Load Dialog](#). You can then change any parameter as desired. You can select  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab to change the drawing parameters of the load.

---

## Load Fields on Onelines

---

Load field objects are used primarily to indicate various quantities associated with load devices. Furthermore, some load field types, which are distinguished by an integrated spin button, may be used to change load device properties.

### Run Mode



For load fields with an associated spin button, clicking on the up/down arrows will change the value of the associated field.

Right clicking on a load field gives you the option to open the [Load Field Dialog](#) or the [Load Information Dialog](#).

### Edit Mode

Simulator offers two options for adding load fields to a oneline in Edit Mode. If you need to enter only a single field, the easier approach may be to choose **Field > Load Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab and then select the load to which you want to add the field. This invokes the [Load Field Dialog](#). Enter the bus number associated with the device (the default is the bus associated to the closest load to the field), the ID field, the total number of digits to show, and the number of digits to the right of the decimal point. Next, select the type of field to show.

The second approach for adding new generator fields entails right-clicking the bus and selecting *Add New Fields Around Load* from the resulting local menu. Please see [Inserting and Placing Multiple Display Fields](#) for more details.

To modify the parameters of an existing load field, position the cursor anywhere on the object and right-click. This brings up the [Load Field Dialog](#). Choose , or  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab to change various display attributes of the field, including its font and background color.

---

## Transmission Line Display Objects

---

Transmission lines are represented on the onelines using multiple segment lines drawn between buses. Transmission lines may be equipped with circuit breakers that can be used to change the line's status. You can also add [pie charts](#) and [line fields](#) to transmission lines to indicate how heavily loaded the line is. The appearance of transmission lines, including line thickness and color, may also be customized.

### Run Mode

Simulator's [animation](#) feature can be used to indicate the magnitude of the flow on the transmission line, either in MW or in terms of the line's percentage loading. You can customize the line flow animation using the *Animated Flows Options* on the [Online Display Options Dialog](#).

Right-clicking on a transmission line displays the line's local menu, from which you can choose to inspect the [Line/Transformer Dialog](#).

### Edit Mode

To add a new transmission line to the case, first select **Network > Transmission Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then place the cursor on the first bus for the transmission line (the "from" bus) and click the left mouse button. Add more segments to the line by moving the cursor and clicking with the left mouse button. To complete adding a new line, place the cursor on the second bus for the line (the "to" bus) and double-click with the left mouse button. This calls up the [Line/Transformer Dialog](#). The "from" and "to" bus numbers are set automatically provided the line starts and ends on existing buses. If there is just one line between the buses, the circuit number should be "1." For multiple lines between buses, you must give each a unique circuit number. Enter the thickness of the lines [in pixels] used to display the transmission line. Enter the per unit (100 MVA base) resistance, reactance, total charging susceptance (that is B not B/2) for the line, and an MVA rating. Select OK to add the line. If you do not want to add the line to the case, select Cancel.

To modify the parameters for an existing line, position the cursor anywhere on the line and right-click. This brings up the [Line/Transformer Dialog](#), which you can use to change various line parameters.

To change the physical appearance of the line (i.e. line color, line thickness, etc.), make use of the tools in the [Formatting](#) ribbon group.

To change the shape of the line, first left-click on the line to select it. This causes handles to appear at each vertex. You can then move any vertex clicking and holding the left mouse button down on the vertex, dragging it to a new location, and then releasing the mouse button. To remove a vertex, hold down the CTRL key and then click the vertex you would like to delete. To add a vertex, hold down the CTRL key and then click on the line where you would like to add a vertex.

---

## Line Fields on Onelines

---


Line field objects are used to show values associated with lines, transformers and dc transmission lines.

### Run Mode

Right clicking on the line field gives you the option to open the [Line Field Dialog](#) or the [Line Information Dialog](#).

### Edit Mode

To enter a new line field, select **Field > Line Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab and select the line, transformer, or dc line to which you want to add the field. The [Line Field Dialog](#) will open. Enter the *near* and *far* bus number associated with the device (the default values for these fields correspond to the device on which you clicked) and the circuit number of the device. The field will display the flow value at the *near* end of the device. Enter the total number of digits to show on the screen and the number of digits to the right of the decimal point. Finally, select the desired field type.

To modify the parameters of an existing line field, position the cursor anywhere on the object and right-click. This again brings up the [Line Field Dialog](#). Select  from the [Formatting](#) ribbon group on the Draw ribbon tab to change many of the line field's display attributes.

Another way to add line fields to the oneline entails right-clicking the line and selecting *Add New Fields Around Line* from the resulting local menu. Please see [Inserting and Placing Multiple Display Fields](#) for more details.

---

## Circuit Breakers on Onelines

---

Circuit breakers are used to open or close transmission lines and transformers. They are also used to place generators, loads and switched shunts in or out of service. By default, closed circuit breakers are shown as solid red squares, while open circuit breakers are shown as a green square outline. The color and filled or unfilled properties of circuit breakers can be modified in the [Online Display Options](#) under Display Object Options.


To change the status of a breaker, left-click on the breaker while in run mode. A circuit breaker directly controls the status of its associated display object. One breaker is shown on the line connecting a generator, load, or shunt to its associated bus. Two breakers are shown by default on transmission lines and transformers. Opening either of the line's circuit breakers opens the transmission line or transformer; you do not have to open circuit breakers at both ends of the line.

Note: Circuit breakers cannot be placed on dc transmission lines.

### Edit Mode

By default, when you add a new display object to a oneline diagram, the necessary circuit breakers are automatically added at each end of the branch.

To add a new circuit breaker to the oneline, select **Indication > Circuit Breaker** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab and then click the line to which you want to add the breaker. The Circuit Breaker Options Dialog will appear with the *from* and *to* bus numbers, and the circuit number automatically set to identify the line you selected. Specify the size for the switch and its initial status, as well as whether it will be anchored to the line so that it will move with the line. Click *OK* to add the circuit breaker, or click *Cancel* to abort the process. To add more circuit breakers to the line, simply repeat this procedure.

To modify the parameters for an existing breaker, position the cursor anywhere on the device and right-click. This invokes the Circuit Breaker Options Dialog, from which you can change many of the breaker's parameters. Additional display settings for the circuit breaker can be accessed by selecting  from the [Formatting](#) ribbon group.

You can toggle the status of Transmission Line and Transformer circuit breakers while in Edit Mode by right-clicking on the associated line or transformer and selecting **Open Line** or **Close Line** from the local menu.

### Run Mode

Circuit Breaker status can be toggled in Run Mode by Left-Clicking on a breaker to place its associated object either in or out of service. If the simulation is currently running, any effects of changing a circuit breaker's status are immediately shown on the oneline.

---

## Line Flow Pie Charts on Onelines

---

The line flow pie charts are used to indicate the percentage MVA, MW, or Mvar loading of a transmission line or a transformer. The degree to which the pie chart is filled shows how close the device is to its limit (provided the device has a nonzero limit). A line flow pie chart becomes completely filled when the device's flow meets or exceeds 100% of its rating.

For pie charts for objects that are not associated with line flows, see [Pie Charts / Gauges](#).

Use the *Pie Chart Options* tab of the [Online Display Options Dialog](#) to customize various attributes of all pie charts. The tab allows you to define a warning level at which the size and color of the pie charts will change to a size and color you specify. The tab also allows you define a limit percent as well as the size and color to which to change the pie charts when their corresponding devices violate their limits. You can also specify whether the pie charts should reveal total power flow (MVA), real power flow (MW), or reactive power flow (MVR). The online display options dialog can be invoked either by selecting **Online Display Options** from the **Online Options** ribbon group on the [Options](#) ribbon tab or by right-clicking on the background of the oneline diagram and selecting *Online Display Options* from the resulting local menu.

Right clicking on the pie chart displays the Line Flow Pie Chart Options Dialog. This dialog allows you to view the *from* and *to bus* numbers and the circuit number of the line/transformer associated with the pie chart. You can change the pie chart's size and the MVA rating of the line/transformer associated with the pie chart.

### Edit Mode

To enter a new line flow pie chart, select **Pies/Gauges > Line Flow Pie Chart** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab [Ref1095648981](#) and click on the line or transformer to which you want to add the pie chart. This opens the Line Flow Pie Chart Options Dialog box. Enter the near and far bus number associated with the device (these fields default to the terminal bus numbers of the device on which you clicked), the circuit number of the device, and the desired size of the pie chart. The field will display the flow value at the *near end* of the device. Enter the size of the device. Select *OK* to insert the line flow pie chart. Otherwise, select **Cancel**.

To modify the parameters of an existing line flow pie chart, position the cursor anywhere on the object and right-click. This again brings up the Line Flow Pie Chart Options Dialog box.

---

## Line Flow Gauges

---

Line flow gauges provide a way to visualize the flow of a transmission line or transformer relative to its thermal rating. A gauge looks very much like a thermometer. As the temperature changes, the height of the mercury in the thermometer moves up and down. One reads the temperature measured by the thermometer by noting the marking that matches the top of the mercury. Line flow gauges in Simulator work the same way. A line flow gauge has two markings on its side, one for the designated minimum flow, and one for the designated maximum flow (the branch's rating). Inside the gauge is a filled region. The default color of the filled region is blue, but this can be changed. When you create the line flow gauge, you associate it with a transmission element, and you specify its fill color and its minimum and maximum flow levels. Once the gauge has been placed on a display, it will reveal changes in its associated branch's flow by varying the height of its filled region. This tool was introduced to provide an alternative to [line flow pie charts](#).

To add a line flow gauge to a display, switch to Edit Mode and select **Pies / Gauges > Old Gauges > Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Click on the oneline diagram where you would like the new line flow gauge to appear. The [Line Flow Gauge Options Dialog](#) will appear. Use this dialog to define the minimum and maximum flows for the line flow gauge, as well as its fill color and whether it should be anchored to its associated transmission element. After you click the "OK" button, the Line Flow Gauge Options Dialog will close, and the new line flow gauge will appear.

Once a gauge has been placed on the oneline, the height of its filled region will change as the flow of its transmission element changes. To modify any of the characteristics of the gauge, such as its key flow levels, fill color, and anchor setting, simply right-click on the line flow gauge to open the Line Flow Gauge Options Dialog again.



---

## Line Flow Gauge Options Dialog

---

The Line Flow Gauge Options Dialog is used to define and configure a [Line Flow gauge](#). A line flow gauge is associated with a particular line and reveals the line's flow relative to specified minimum and maximum thermal flow levels. The height of the colored column in the line flow gauge indicates the line's flow relative to these markings.

The dialog has the following controls:

### Number, Name, and Find

Use the **Number** and **Name** dropdown boxes, in addition to the **Circuit** field, to identify the terminal buses to which the branch desired corresponds. It may be more convenient to press the **Find** button to open the [Find Dialog](#), which allows you to specify the line by either bus names or numbers using wildcards. When you first open the Line Flow Gauge Dialog, the bus names and numbers will correspond to the transmission line object that was closest to the point where you clicked.

### Minimum and Maximum

Use these two spin edit boxes in the **MW Rating values** group box to specify the minimum and maximum flow levels. These settings determine where on the gauge its two markings will be drawn.

The Minimum and Maximum value will be taken from the Limit Monitoring settings for the current transmission line, unless you uncheck the option "*Set Limits According to Current Limit Monitoring Settings*".

### Anchored

A line flow gauge is said to be [anchored](#) if, when you move its associated line object, the gauge moves with it. Check the Anchored check box to ensure that the gauge will move with its associated line. Otherwise, when you move its associated line object, the gauge will stay in its current position.

### OK, Help, and Cancel

Click OK to finalize your settings. This will create a new line flow gauge object if you are trying to create one from scratch, or it will modify the appearance and settings of an existing one if you have chosen to modify one that has already been defined. Click Cancel to dispose of your changes.

---

## DC Transmission Line Display Objects

---

DC transmission lines are represented on the onelines using multiple segment lines drawn between two buses. [Line fields](#) are often placed close to dc transmission lines on the oneline to indicate the power flow through the device.

Note that, unlike ac transmission lines, dc transmission lines cannot be equipped with circuit breakers.

### Run Mode


Simulator's [animation](#) feature can be used to indicate the magnitude of the flow on the dc transmission line, either in MW or in terms of the line's percentage loading. You can customize the line flow animation using the *Animated Flows Options* on the [Online Display Options Dialog](#).

Right-clicking on a dc transmission line displays the line's local menu, from which you can choose to inspect the [DC Transmission Line Dialog](#).

### Edit Mode

To add a new dc transmission line to the case, first select **Network > DC Transmission Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then place the cursor on the bus you desire to be the rectifier bus for the line and click the left mouse button. Add more segments to the line by moving the cursor and clicking with the left mouse button. To complete the new line, place the cursor on the second bus for the line, which will serve as the inverter bus, and double-click the left mouse button. This calls up the [DC Transmission Line Dialog](#). If you successfully selected the rectifier and inverter buses, their numbers will be automatically filled in for you when the dialog opens.

The DC Transmission Line Record Dialog has four separate pages: Line Parameters, Rectifier Parameters, Inverter Parameters, and Actual Flows. The separate pages can be accessed using the tabs shown at the top of the dialog. These pages are used to set the modeling parameters associated with the dc lines.

To modify the parameters for an existing dc transmission line, position the cursor anywhere on the line and right-click. This will provide access to the corresponding [DC Transmission Line Dialog](#), from which you can modify any of the dc line's parameters. Select  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab to change the color and/or line thickness of the dc line.

To change the shape of the line, first left-click on the line to select it. This causes handles to appear at each vertex. You can then move any vertex by clicking and holding the left mouse button down, dragging the vertex to a new location, and releasing the mouse button. To remove a vertex, hold down the CTRL key and then click the vertex you would like to delete. To add a vertex, hold down the CTRL key and then click on the line where you would like to add a vertex.

---

## Multi-section Line Display Objects

---

Multi-section lines are represented on the onelines using a line segment with intermediate or "dummy" bus representations. These objects are not the same as a normal transmission line. They do display animated flows in run mode, but do not have pie charts, circuit breakers, or text fields currently associated with the objects.

### Run Mode

Simulator's [animation](#) feature can be used to indicate the magnitude of the flow on the transmission line, either in MW or in terms of the line's percentage loading. You can customize the line flow animation using the *Animated Flows Options* on the [Online Display Options Dialog](#).

Right-clicking on a multi-section line displays the [Multi-section Line Information Dialog](#).

### Edit Mode

Unlike transmission lines and transformers, multi-section lines CANNOT be inserted in a case by adding the multi-section line graphically. Therefore, when inserting a multi-section line object, the data for the object must already exist in the case. A list of currently defined multi-section lines is found in the model explorer under **Aggregations > Multi-Section Lines**. Right-clicking and selecting **Insert...** in the table that opens will allow you to add new multi-section line information via the [Multi-Section Line Information](#) dialog.

To insert the multi-section line object, first select **Network > MS Transmission Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then place the cursor on the first bus for the transmission line (the "from" bus) and click the left mouse button. Add more segments to the line by moving the cursor and clicking with the left mouse button. To complete adding a new line, place the cursor on the second bus for the line (the "to" bus) and double-click with the left mouse button. Completing the multi-section line opens the [Multi-section Line Information](#) dialog. At the bottom of this dialog are display options for the line, such as line pixel thickness, symbol size (for the series capacitor representations, if any,) anchored, and options for drawing the buses.

To change the shape of the line, first left-click on the line to select it. This causes handles to appear at each vertex. You can then move any vertex clicking and holding the left mouse button down on the vertex, dragging it to a new location, and then releasing the mouse button. To remove a vertex, hold down the CTRL key and then click the vertex you would like to delete. To add a vertex, hold down the CTRL key and then click on the line where you would like to add a vertex.

---

## Transformer Display Objects

---

Transformers are represented as transmission lines with two opposing coils drawn on one of the segments. The transformer's line thickness, color, and symbol segment can be customized using the Edit Mode. Optionally, [circuit breakers](#) and [pie charts](#) can be placed on the transformer. Clicking on the circuit breakers changes the status of the transformer.

[Line fields](#) are often placed close to transformers on the oneline to indicate the power flow through the device. [Transformer fields](#) are often placed close to transformers on the oneline to indicate and control their tap positions. See [Transformer Modeling](#) for details on modeling either LTC or phase shifting transformers.

### Run Mode

Simulator's [animation](#) feature can be used to indicate the magnitude of the flow through the transmission line, either in MW or in terms of the transformer's percentage loading. You can customize the line flow animation using the *Animated Flows Options* on the [Oneline Display Options Dialog](#).

Right-clicking on a transformer displays the line's local menu, from which you can choose to inspect the [Line/Transformer Dialog](#).

### Edit Mode

New transformers are inserted in much the same way as transmission lines. To add a new transformer to the case, first select **Network > Transformer** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then place the cursor on the first bus for the transformer (the *from bus*) and click the left mouse button. Add more segments to the transformer by moving the cursor and clicking with the left mouse button. To complete the new transformer, place the cursor on the transformer's other terminal (the *to bus*) and double-click with the left mouse button. This calls up the [Line/Transformer Dialog](#). The *from* and *to* bus numbers for the transformer should have been set automatically. If there is just one transformer between the buses, the circuit number should be "1." For multiple transformers between buses, you must give each a unique circuit number. Enter the thickness of the lines [in pixels] used to display the transformer, the number of the line segment in which you would like the transformer symbol drawn, and the size of the transformer symbol.

Enter the per unit (100 MVA base) resistance, reactance and charging susceptance for the transformer, and an MVA rating. Enter the off-nominal tap ratio and the phase shift angle in degrees. (For a transformer without tap or phase control, the off-nominal tap should be 1.0 and the phase shift angle should be 0 degrees.)

Select the appropriate Automatic Control Option. If the transformer does not have tap control, select *No Automatic Control* (this is the default). Select *AVR* (Automatic Voltage Regulation) if the transformer changes its tap ratio to control the voltage at user specified regulation bus. Select *Reactive Power Control* if the transformer changes its tap ratio to control the reactive power through the transformer. Finally, select *Phase Shift Control* if the transformer changes its phase shift to control the MW flow through the transformer. If you need any of the last three options, select the Automatic Control Options button to set the parameters associated with the automatic control.

For AVR control, enter the number of the bus whose voltage is to be controlled, the allowable range for the controlled voltage (in per unit), the minimum and maximum tap ratios (typical values are 0.9 and 1.1), and the step size for the discrete changes in the tap ratio (typical value is 0.00625).

For reactive power control, the control variable is always the reactive power measured at the *from bus* (i.e., the tapped side) of the transformer. Positive flow is assumed to be going through the transformer to the *to bus*. Enter the minimum and maximum allowable flows, the minimum and maximum tap ratios (typical values are 0.9 and 1.1), and the step size for the discrete changes in the tap ratio (typical value is 0.00625).

For phase shift control, the MW flow through the transformer is the controlled value. Enter the bus number of the terminal whose flow is controlled, the allowable range for the controlled flow (positive flow is assumed to be into the transformer at the terminal entered in the previous field), the minimum and maximum phase angles (typical values are  $-30^\circ$  and  $30^\circ$ ), and the step size in degrees (typical values are between  $1^\circ$  and  $2^\circ$ ).

Select **OK** to save the values and return to the Transformer Options Dialog; otherwise select **Cancel**.

If you would like the transformer to be initially modeled as being on automatic control at the start of the case, select the *Automatic Control Active* checkbox.

If you do not want to add the transformer to the case, select Cancel.

To modify the parameters for an existing transformer, position the cursor anywhere on the device and right-click. This brings up the local menu from which you can choose to view the [Line/Transformer Dialog](#). Use the Line/Transformer Dialog to adjust many of the transformer's electrical properties. The [Formatting](#) ribbon group provides tools to change the transformer's color and/or line thickness.

See [Transformer Modeling](#) for details on modeling either LTC or phase shifting transformers.

---

## Transformer Fields on Onelines

---

Transformer fields are used to show field values specific to transformers, such as tap position, phase angle, and more.

### Run Mode

If there is a spin control integrated with the field, you can click on the spinner to change the field's value by the associated *Delta Per Mouse Click*.

Right clicking on the transformer field gives the option to display the [Transformer Field Dialog](#) or the [Line/Transformer Information Dialog](#).

### Edit Mode

To enter a new transformer field, select **Field > Transformer Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab, and then click on the transformer object to which you want to add the new field. This calls up the [Transformer Field Dialog](#). Enter the *from* and *to* bus numbers associated with the device (the default values for these fields correspond to the transformer on which you clicked), and the circuit number of the device. Enter the total number of digits that the field should display, as well as the number of digits to the right of the decimal point. Finally, specify what the field should display: the off-nominal tap ratio, the off-nominal tap position, or the phase shift angle in degrees.

To modify the parameters of an existing transformer field, position the cursor anywhere on the object and right-click to bring up the [Transformer Field Dialog](#). Use the tools of the [Formatting](#) ribbon group to change various display attributes for the transformer field, including its font and background color.

---

## Series Capacitor Display Objects

---

Series capacitors are represented as transmission lines with two opposing parallel bars drawn on one of the segments. The series capacitor's line thickness, color and symbol segment can be customized using the Edit Mode. If the series capacitor branch is operating, but the series capacitor status is set to **Bypassed**, a low impedance segment will be drawn around the series capacitor symbol to indicate the capacitor has been bypassed. The capacitor status of **Bypassed** or **In Service** can be toggled in run mode if the Series Capacitor Status field is displayed on the oneline diagram. When a left-click is registered on the Series Capacitor Status field when in Run Mode, the capacitor status is toggled. Note that this is not the same as the overall branch status of **Open** or **Closed**.

Optionally, [circuit breakers](#) and [pie charts](#) can be placed on the series capacitor. Clicking the circuit breakers changes the branch status of the series capacitor.

### Run Mode

Simulator's [animation](#) feature can be used to indicate the magnitude of the flow through the series capacitor, either in MW or in terms of the transformer's percentage loading. You can customize the line flow animation using the *Animated Flows Options* on the [Online Display Options Dialog](#).

Right-clicking on a transformer displays the line's local menu, from which you can choose to inspect the [Line/Transformer Dialog](#).

### Edit Mode

New series capacitors are inserted in much the same way as transmission lines. To add a new series capacitor to the case, select **Network > Series Capacitor** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then place the cursor on the first bus for the series capacitor (the *from bus*) and click the left mouse button. Add more segments to the series capacitor by moving the cursor and clicking with the left mouse button. To complete the new series capacitor, place the cursor on the series capacitor's other terminal (the *to bus*) and double-click with the left mouse button. This calls up the [Line/Transformer Dialog](#). The *from* and *to* bus numbers for the series capacitor should have been set automatically. If there is just one branch between the buses, the circuit number should be "1." For multiple branches between buses, you must give each a unique circuit number. Enter the thickness of the lines [in pixels] used to display the series capacitor, the number of the line segment in which you would like the series capacitor symbol drawn, and the size of the series capacitor symbol.

Enter the per unit (100 MVA base) resistance, reactance and charging susceptance for the series capacitor, and an MVA rating. On the Series Capacitor tab, check the box labeled **Is Series Capacitor** to indicate that the branch model is a series capacitor device.

If you do not want to add the series capacitor to the case, select Cancel. Otherwise click **OK** to add the series capacitor to the case.

To modify the parameters for an existing series capacitor, position the cursor anywhere on the device and right-click. This brings up the local menu from which you can choose to view the [Line/Transformer Dialog](#). Use the Line/Transformer Dialog to adjust many of the series capacitor's electrical properties.

See [Series Capacitor Information](#) for more details on modeling series capacitors.

---

## Series Capacitor Fields on Onelines

---

Series capacitor fields are used to show field values specific to series capacitors.

### Run Mode

If the Series Capacitor field for capacitor status is displayed, the capacitor status can be toggled in run mode when you left-click on the field.

Right clicking on the series capacitor field gives the option to display the [Series Capacitor Field Dialog](#) or the [Line/Transformer Information dialog](#).

### Edit Mode

To enter a new series capacitor field, select **Field > Series Capacitor Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab, and then click on or near the series capacitor object for which you want to add the new field. This calls up the [Series Capacitor Field Dialog](#). Enter the *from* and *to* bus numbers associated with the device (the default values for these fields correspond to the series capacitor on which you clicked), and the circuit number of the device. Enter the total number of digits that the field should display, as well as the number of digits to the right of the decimal point. Finally, specify what the field should display: the capacitor status or the series capacitance.

To modify the parameters of an existing series capacitor field, position the cursor anywhere on the object and right-click to bring up the [Series Capacitor Field Dialog](#).



---

## Switched Shunt Display Objects

---

Switched shunts are either capacitors that supply reactive power to the system or reactors that absorb reactive power. Simulator represents switched shunts as a number of blocks of admittance that can be switched in a number of discrete steps or over a continuous range. Switched shunt display objects come equipped with a circuit breaker that indicates the shunt's status. If the switched shunt is closed, the circuit breaker appears as a filled red square. If the switched shunt is open, the circuit breaker appears as a green square outline. To change the status of the switched shunt, click the corresponding circuit breaker.

[Switched shunt fields](#) are often placed next to switched shunts to indicate the amount of reactive power supplied by the device. For switched shunts with such a field, you can manually increase the reactive power supplied by the device (provided its control mode is discrete) by clicking on the up-arrow associated with the device's reactive power field. Likewise, you can decrease the reactive power supplied by the device by clicking on the down-arrow. To make the up/down arrows visible, set the *Delta per Mouse Click* on the [switched shunt field](#) to a nonzero value.

Right-clicking on the switched shunt displays the [Switched Shunt Dialog](#). Use the Switched Shunt Dialog to inspect or modify the model of the switched shunt.

You can add a new switched shunt to the case in Edit Mode. Select **Network > Switched Shunt** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then click the bus where you would like to attach the device. The [Switched Shunt Dialog](#) will appear. The bus number is automatically determined from the bus to which you attached the capacitor. Enter the size, the thickness of the pen [in pixels] used to draw the device, and its orientation. The Nominal Mvar field gives the amount of reactive power the device would supply if its terminal voltage were 1.0 per unit. The Control Mode field determines whether the switched shunt has a fixed value or will vary discretely or continuously within its operating limits to maintain its terminal voltage within the voltage range specified in the Voltage Regulation field.

The amount of shunt admittance is specified in the Switched Shunt Blocks table. The columns in this field correspond to different blocks of admittance. The first row indicates the number of steps in each block, and the second row gives the amount of nominal Mvars per step. The switched shunts are always switched in the order specified in this field.

Select OK to add the device. If you do not want to add the switched shunt to the case, select Cancel.

To modify the parameters for an existing switched shunt, position the cursor on the device and right-click. This again brings up the [Switched Shunt Dialog](#). You can then change any parameter of the switched shunt. Make use of the tools in the [Formatting](#) ribbon group to change the color and/or line thickness.

To delete an existing switched shunt, use the **Cut** or **Delete** commands in the [Clipboard](#) ribbon group.

---

## Switched Shunt Fields on Onelines

---

Switched Shunt field objects are used primarily to indicate various quantities associated with switched shunt devices. Furthermore, some switched shunt field types, which are distinguished by an integrated spin button, may be used to change switched shunt device properties.

### Run Mode

For switched shunt fields with an associated spin button, clicking on the up/down arrows will change the value of the associated field.

Right clicking on a generator field gives you the option to open the [Switched Shunt Field Dialog](#) or the [Switched Shunt Information Dialog](#).

### Edit Mode

Simulator offers two options for adding switched shunt fields to a oneline in Edit Mode. If you need to enter only a single field, the easier approach may be to choose **Field > Switched Shunt Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab, and then select the switched shunt to which you want to add the field. This invokes the [Switched Shunt Field Dialog](#). Enter the bus number associated with the device (the default is the bus associated to the closest switched shunt to the field), the ID field, the total number of digits to show, and the number of digits to the right of the decimal point. Next, select the type of field to show.

The second approach for adding new switched shunt fields entails right-clicking the bus and selecting *Add New Fields Around Switched Shunt* from the resulting local menu. Please see [Inserting and Placing Multiple Display Fields](#) for more details.


To modify the parameters of an existing switched shunt field, position the cursor anywhere on the object and right-click. This brings up the [Switched Shunt Field Dialog](#). Make use of the tools on the [Formatting](#) ribbon group to change various display attributes of the field, including its font and background color.

---

## Interface Display Objects

---

Interface display objects are used on the onelines to visualize the flow of power through an [interface record](#). Interface records are used to show the net real power (MW) flow on a group consisting of one or more of the following devices: 1) transmission lines and/or transformers, 2) total tie-lines between two adjacent areas, and 3) total tie-lines between two adjacent zones. Only area-area and zone-zone interface records can be displayed using interface display objects.

Interfaces, like transmission lines, are represented as multi-segment lines, except that they may be drawn between [area/zone objects](#) in addition to buses. Drawing interface display objects involves the same steps as drawing transmission lines and transformers. The line thickness and color of interface objects may be customized by selecting  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab.

[Interface Fields](#) and [Interface Pie Charts](#) are often placed close to or on the interface to indicate the power flow through the device.

### Run Mode

When animation is active, the flow of the arrows on the interface object may represent either the MW flow through the interface or the currently calculated power transfer distribution factor ([PTDF](#)) pertaining to that interface. You can customize the appearance of the animated flows using the *Animated Flows Tab* of the [Online Display Options Dialog](#) available using the local menu.

Right-click on the interface to view the [Interface Dialog](#) for the interface.

### Edit Mode

The quickest method of inserting new interface objects is to use the **Auto Insert > Interfaces** from the [Quick Insert](#) ribbon group on the [Draw](#) ribbon tab. Please see [Automatically Inserting Interfaces](#) for details.

To manually add a new interface object to the case, first select **Aggregation > Interface** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then place the cursor on the starting location for the interface (usually an area/zone object or a bus object) and left-click. Add more segments to the interface by moving the cursor and the left-clicking at the end of the segment. To finish adding an interface, place the cursor on the terminal object for the interface and double-click. This then calls up the [Interface Dialog](#). Either select an existing interface, or define a new interface (see [Interface Dialog](#) for details).

Interfaces can be [anchored](#) to either area/zone objects or bus objects.

To change the shape of the interface, first left-click on the object to select it. This causes handles to appear at each vertex. You can then move any vertex by dragging it with the left mouse button down. To remove a vertex, hold down the CTRL key and then click the vertex you would like to delete. To add a vertex, hold down the CTRL key and then click on the interface where you would like to add the vertex.

---

## Automatically Inserting Interface Display Objects

---

The Automatic Insertion of Interfaces Dialog is used to automatically insert [interface objects](#) on the oneline diagram between existing [area/zone display objects](#). [Interface pie chart objects](#) can also be inserted as part of this process. The automatic insertion of interface display objects greatly accelerates the construction of interface diagrams, which are particularly useful for animating the results of [PTDF](#) calculations. Area-area and zone-zone interface records can be displayed using interface display objects as well as interfaces comprised only of transmission branches.

Note that inserting interface objects on a oneline diagram does NOT add interface objects to the load flow case. The interface definitions need to be added to the load flow case prior to automatically inserting the graphical interface objects on a oneline diagram. The quickest way to add area to area or zone to zone interface definitions in the load flow case is to [automatically insert the definitions](#) in the interface case information display.

Once you have the interface definitions defined in the case, you can automatically insert the interface objects on a diagram using the following procedure:

- On a oneline diagram (either an already open diagram or a brand new one created by choosing **New Oneline** from the [application button](#)), place [area/ zone objects](#) at the desired locations if inserting area-area or zone-zone interface records. Otherwise, the terminal buses for the branch elements in the interface should be inserted at the desired locations on the oneline diagram.
- Go to the [Draw](#) ribbon tab and choose **Auto Insert > Interfaces** on the [Quick Insert](#) ribbon group.
- Check the **Insert Pie Charts on Interfaces** box to automatically insert interface pie chart objects when the interfaces are inserted. If this option is selected, change the **Default Size of Interface Pie Charts** to specify their size.
- Choose the type of interfaces to insert. **Area to Area or Zone to Zone Interfaces** and **Line/Transformer Interfaces** can be inserted. Area-area and zone-zone interfaces will be drawn between the respective area and zone objects displayed on the oneline. Line/transformer interfaces will be drawn based on the average location of the terminal buses of all transmission lines comprising the interface. If inserting line/transformer interfaces, the option **Minimum Length of Line/Transformer Interfaces** can be set to prevent interfaces that are too short from being inserted.
- Select **OK** to insert the new oneline objects. New interface objects are automatically inserted based on the selected criteria for any corresponding interface record that is not already represented.

Note that you can do this automatic insertion as often as you like. The **Number of Interfaces Not Shown** field indicates how many interfaces still need to be added to the diagram to represent all defined area-area, zone-zone, or line/transformer interfaces. It is a read-only field.

---

## Interface Fields on Onelines

---

Interface field objects are used to show values associated with [interface records](#).

### Run Mode

Right clicking on the interface field displays the [Interface Field Dialog](#).

### Edit Mode

To enter a new interface field, select **Field > Interface Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab, and then click on the background of the online diagram where you want the field placed. This calls up the [Interface Field Dialog](#). Enter the name of the interface, the total number of digits desired in the field, and the type of field.

To modify the parameters of an existing interface field, position the cursor anywhere on the object and right-click. This again brings up the Interface Field Dialog.

---

### **InterArea Flow Options Dialog**

---

This dialog is outdated in PowerWorld Simulator version 5 and later. See help on the [Interface Field Information Dialog](#) for the updated dialog.

When viewing an area diagram containing inter-area objects created using PowerWorld Simulator version 4.2 or older, this dialog allows you to set text fields displaying either the actual or scheduled MW flow on the inter-area object.

---

## Interface Pie Charts on Onelines

---

Interface pie charts are used to graphically show the percentage loading on an [interface record](#). The amount of shaded region of the pie chart indicates how close the interface is to its limit (provided the interface has a nonzero limit). The appearance of the interface pie charts, including their color and the ability to automatically change size based upon loading level, can be customized on the [Oneline Display Options Dialog](#).

Right-clicking on the interface pie chart displays the [Interface Pie Chart Dialog](#). This dialog can be used to customize the size of the pie chart, or change the interface's limit.

### Edit Mode

To enter a new interface pie chart, select **Pies/Gauges > Interface Pie Chart** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then, click the left mouse on the interface object with which you want to associate the pie chart. This calls up the [Interface Pie Chart Dialog](#), which is used to customize the appearance of the pie chart.

---

## Loading NERC Flowgates

---

This command reads flowgates from an Excel file and inserts them as interface records. The format for this file should be similar to the files found at <http://www.nerc.com/~oc/dfwg.html>. To access this file you must have a NERC-supplied username and password. At the time of writing we have not seen an official description of the format -- currently Simulator just mimics the format found in this file. Flowgates are used by NERC (under Policy 9) as proxies for transmission limitations and transmission service usage on the interconnected electric power network. Simulator models flowgates using the interface records. Interface records MAY include contingency elements.

The **Load NERC Book of Flowgates from Excel** option is available from the [local menu](#) of the [Interface Records](#) display.



---

## Saving NERC Flowgates

---

This command writes all the interfaces to an Excel spreadsheet using the NERC flowgate format. See the files found at <http://www.nerc.com/~oc/dfwg.html> for an example of the NERC format. To access this file you must have a NERC-supplied username and password.

The **Send NERC Book of Flowgates to Excel** option is available from the **Save As** option of the [local menu](#) of the [Interface Records](#) display.

---

## Links to Onelines and Auxiliary Files

---

### Run Mode

#### Online Links

Online links are one of the mechanisms used in Simulator to allow you to view multiple online diagrams on the same screen. By default, the online links are shown as blue rectangles surrounding a text identifier of the linked online diagram. Double-clicking the left mouse button anywhere within the online link object will display the linked online diagram, even if it has not already been opened.

The ability to associate multiple online diagrams with a single case may prove particularly helpful when dealing with large cases. If a single online diagram is used to display a case having many buses, the diagram may become cluttered, and it will be difficult to analyze the case in sufficient detail. The ability to associate additional onelines with the case and to call up those additional onelines using one-links can significantly enhance your view of the system.

Note that you can also use the **Open Online** command from the [Application Button](#) to open any online diagram (\*.pwd) file directly. See [Opening a Online Diagram](#) for further details.

#### Auxiliary File Links

Auxiliary (\*.aux) file links are one of the mechanisms used in Simulator to allow you to load auxiliary files by simply clicking on a online object. By default, auxiliary file links are shown as blue rectangles surrounding a text identifier of the linked auxiliary file. Double-clicking the left mouse button anywhere within the auxiliary file link object will load the associated auxiliary file.

Note that you can also use the **Load Auxiliary File** command from the [Application Button](#) to open any auxiliary file (\*.aux) directly. See [Auxiliary Files](#) for further details.

### Edit Mode

#### Online Links


To add a new online link to a online diagram, first select **Background > Online Link** from the Individual Insert ribbon group on the [Draw](#) ribbon tab. Then, click the left mouse button at the location where you would like to display the new link. The Online Link Options Dialog box will open, asking you to enter the name of the file that stores the linked online and a caption for the clickable online link object. Note that you can enter simply the name of the online file in this location. Simulator will look in the same directory as the case file by default. However, you can also specify additional directory locations to search for online diagrams by opening the [Online Display Options Dialog](#) (via **Online Display Option** from the **Online Options** ribbon group on the [Options](#) ribbon tab, or from the [Active](#) ribbon group on the [Onelines](#) ribbon tab), switching to *Display Options* on the left, and using the *Edit Online Browsing Path* option to add additional search locations for online diagrams.

#### Auxiliary File Links

To add a new auxiliary file link to a online diagram, first select **Background > Auxiliary File Link** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then, click the left mouse button at the location where you would like to display the new link. The *Online and Auxiliary Link Options Dialog* will open, asking you to enter the name of the file that stores the linked auxiliary file and a caption for the clickable auxiliary file link object. Note

that you can simply enter the name of the auxiliary file in this location. Simulator will look in the same directory as the case file.

### **Modifying Existing Links**

To modify the parameters of an existing online or auxiliary file link, position the cursor anywhere on the object and right-click. This invokes the *Online Link Options Dialog*, allowing you to change the identity of the linked file and the caption. Select  from the Formatting ribbon group on the [Draw](#) ribbon tab to modify various display attributes of the online link, including font size and background color.

---


## Document Links on Onelines

---

Just as you can link to other oneline displays from a oneline diagram using [online link objects](#), you can also link to documents and data on the world-wide web using Document Link Objects. When you click on Document Link Objects in Run Mode, your system's default browser will be launched to retrieve the linked URL address. This feature is not only for web URL's, though. Any file can be linked and its associated application will automatically open. This means that presentations, documents and spreadsheets can also be linked.

In addition to linking to objects online via a URL, it's also possible to put in a local file name and have that open when the document link is clicked. The behavior is the same as if you were to click on the Windows Start button, then select Run.., then type in the file name.

To add a Document link to a oneline diagram, select **Background > Document Link** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Then, click the left mouse on the oneline diagram at the position where you would link to insert the Document link. The Document Link Options Dialog box will appear, asking you to specify the world-wide web address to which to link as well as a clickable caption to display on the oneline diagram. Enter the requested information and press *OK* to add the Document link object to the oneline.

To modify the caption or address for a Document link object, right-click anywhere on its text. Specify its new parameters in the Document Link Options Dialog and press *OK*. You may also modify various aspects of its appearance, including the font size, by choosing  from the [Formatting](#) ribbon group.

Note that Document links are active only in Run Mode. Clicking on a Document link object from the Edit Mode will have no effect other than to select the object for placement, formatting, etc.

---

## Background Lines on Onelines

---

The background of an oneline diagram can display added lines, polylines, and filled polygons among other items.

### Edit Mode

To add a new background line, first go to the [Draw](#) ribbon tab and select **Background > Background Line** from the [Individual Insert](#) ribbon group. Then to draw a series of straight line segments on the background, follow these steps:

- Position the mouse cursor where you want the series to begin and click and release the left mouse button.
- Move the mouse to the desired termination point of the first line segment. A straight segment will follow your mouse movements. Click and release the left mouse button to complete the line segment and prepare for drawing the next line segment, or double-click if this line segment is the last segment you wish to draw.

To draw a freehand shape rather than a series of straight line segments, click and hold the left mouse button where you would like the freehand shape to begin and drag the mouse to trace the shape you desire (while holding the left mouse button down). Release the left mouse button to complete the section of the freehand shape you have been drawing. At this point, you can add either another freehand section or a straight line segment. When you have finished drawing in the background, double click the mouse button.

Note that background display objects composed of straight line segments display significantly faster than lines drawn freehand. Lines draw freehand (holding down mouse button) leave a vertex point at every point on the screen, where a line composed of straight-line segments (left-clicking only where you want a vertex point) takes considerably less effort for the PC to draw.

To change the color, line thickness, and fill color associated with the line, make use of the tools in the [Formatting](#) ribbon group.

To change the shape of the line, first left-click on the line to select it. This causes handles to appear at each vertex. You can then move any vertex by holding the left mouse button down and dragging the vertex to a new location. To remove a vertex, hold down the CTRL key and then click the vertex you would like to delete. To add a vertex, hold down the CTRL key and then click on the line where you would like to add a vertex. Note that freehand lines are nothing more than a continuous series of vertices.

---

## Background Rectangles on Onelines

---

The background of a oneline diagram can display rectangles among other items.

### Edit Mode

To add a new rectangle, first go to the [Draw](#) ribbon tab and select **Insert > Background Rectangle** from the [Individual Insert](#) ribbon group. Position the cursor where you would like to place the upper left-hand corner of the rectangle and click with the left mouse button. A rectangle having the default size is inserted. Drag the rectangle's resizing handles to resize/reshape the rectangle.

To resize or reshape an existing rectangle, click on it to select it. The resizing handles will appear, which you can then drag to reshape or resize the rectangle.

To change the color, line thickness or fill color of the rectangle, first select the rectangle by clicking on it on the diagram, and then use the tools in the [Formatting](#) ribbon group on the [Draw](#) ribbon tab, or right-click on it and select **Format Background Rectangle** from the local menu.

---

## Background Ellipses on Onelines

---

The background of a oneline diagram can display ellipses among other items.

### Edit Mode

To add a new ellipse, first go to the [Draw](#) ribbon tab and select **Background > Background Ellipse** from the [Individual Insert](#) ribbon group. Then position the cursor where you would like to place the upper left-hand corner of the ellipse and click with the left mouse button. An ellipse having the default size and shape is inserted. Drag the ellipse's resizing handles to resize/reshape it.

To resize or reshape an existing ellipse, click on it to select it. The resizing handles will appear, which you can then drag to reshape or resize the ellipse.

Right-click on the ellipse and choose **Format Background Ellipse**, or select the ellipse and use the tools in the [Formatting](#) ribbon group found on the [Draw](#) ribbon tab to change the color, line thickness and fill color of the ellipse.

---

## Background Pictures on Onelines


---

The background of an oneline diagram can display a variety of objects, including lines, filled polygons, and even pictures. The latter will be discussed in this section.

### Edit Mode

Simulator can insert bitmaps, jpegs, enhanced and standard metafiles, and icons from files into the oneline diagram. These pictures may either serve as a background or appear above other objects on the oneline.

To add a picture object to the oneline diagram, go to the [Draw](#) ribbon tab and select **Background > Picture** from the [Individual Insert](#) ribbon group. Then, click the oneline diagram at the point where you would like the top left corner of the graphic to be placed. The Open Picture Dialog will open, asking you to select the graphic file that you want to insert. When you select a file from the dialog, the dialog displays a preview image so that you can be sure that you are selecting the right file. When you have identified the file to insert, click *OK*. The image will then appear on the oneline. It may be resized by dragging its resizing handles.

By default, pictures are inserted at the middle stack level. Thus, they hide most other oneline display objects. To change the stack level of the picture, click it to select it, and then select the Levels/Layers button  found in the [Formatting](#) ribbon group of the [Draw](#) ribbon tab. From the *Levels/Layers Tab*, select the stack level of your choice. The *Base* stack level places the picture below all other oneline display objects, while the *Top* stack level will cause the picture to obscure all other display objects.

To resize or reshape an existing picture object, click on it to select it and then drag its resizing handles.



---

## Background Lines Dialog

---

The Line Options Dialog is available while in edit mode by right-clicking on a background line and selecting **Open Dialog...** This dialog contains options for formatting the selected background line. Many of the options listed in this dialog can also be updated on the [Format](#) dialog.

### Line Thickness

This field gives the thickness of the line in pixels.

### Line Color

This field shows the current line color. To change the line's color, click on the box displaying the current color or click the **Change** button next to the line color. This will display the Color Dialog. Select the desired color and select OK to change the color.

### Use Background Fill

When this option is checked, the selected Fill Color will be used to fill the background of the line.

### Fill Color

This field shows the current fill color. To change the fill color, click on the box that displays the current fill color or click the **Change** button. This will display the Color Dialog. Select the desired color and select OK to change the color.

### Immobile

When this option is checked, the background line will be forced to stay in the same position and cannot be moved by dragging it with the mouse.

### List of Vertices

This grid lists the x,y coordinates for the vertices of the line in the order in which the line is drawn. The coordinates will be displayed as longitude,latitude if the [option to show coordinates in longitude,latitude](#) is selected on the [Online Display Options](#) dialog and a valid map projection is in use. The values for the coordinates can be changed by entering new values in the appropriate position in the grid.

### Shift X/Y Values

These options provide a means of updating all of the x and/or y values. The **X Shift Value** is the offset by which all of the x values will be shifted. The **Y Shift Value** is the offset by which all of the y values will be shifted. The resulting values will be the original values plus the value entered for the shift. The **Shift All Values** button must be selected for the values entered in the shift fields to be applied to the x,y values. After this button is selected, the coordinates shown in the grid will be updated with the new values.

### Scale X/Y Values

These options provide a means of updating all of the x and/or y values. The **X Scale Value** is the value by which all of the x values will be scaled. The **Y Scale Value** is the values by which all of the y values will be scaled. The resulting values will be the original values multiplied by the value entered for the scale. The **Scale All Values** button must be selected for the values entered in the scale fields to be applied to the x,y values. After this button is selected, the coordinates shown in the grid will be updated with the new values.

### OK, Save and Update Display, and Cancel

**OK** saves the changes and closes the dialog. **Save and Update Display** saves the changes and updates the display without closing the dialog. **Cancel** closes the dialog without saving any changes.

---

## Converting Background Lines

---

Background lines can be converted to other objects by splitting, merging, or converting to power system objects. To convert a background line, right-click on a selected background line on the oneline display and the local menu will be displayed with the conversion options. Most options are available if only a single background line has been selected and no other objects are selected. The following options are available for converting a background line:

### Split Background Line

A background line can be split at a vertex or anywhere along the line. Click the point on the line where the split should be made and then right-click and select the **Split Background Line...** option from the local menu. Two background lines will result. This option is available only if exactly one background line has been selected and no other objects have been selected.

### Merge Background Lines

This option is only available if exactly two background lines have been selected. The two selected background lines will be merged at the two closest ends and a single background line will result.

### Convert to ac Transmission Line

After selecting this option, the [Line/Transformer Information dialog](#) will be displayed. The dialog options can be set the same as they would be set when inserting an ac transmission line from the Insert menu. This option is available only if exactly one background line has been selected and no other objects have been selected.

### Convert to Bus

After selecting this option, the [Bus Information Dialog](#) will be displayed. The dialog options can be set the same as they would be set when inserting a bus from the Insert menu. This option is available only if exactly one background line has been selected and no other objects have been selected.

### Convert to Substation

After selecting this option, the [Substation Information Dialog](#) will be display. The dialog options can be set the same as they would be set when inserting a substation from the Insert menu. This option is available only if exactly one background line has been selected and no other objects have been selected.

---

## Converting Background Ellipses

---

Background ellipses can be converted to power system objects. To convert a background ellipse, right-click on a selected background ellipse on the oneline display and the local menu will be displayed with the conversion options. The following options are available for converting a background ellipse:

### Convert to Bus

After selecting this option, the [Bus Information Dialog](#) will be displayed. The dialog options can be set the same as they would be set when inserting a bus from the Insert menu. This option is available only if exactly one background ellipse has been selected and no other objects have been selected.

### Convert to Substation

After selecting this option, the [Substation Information Dialog](#) will be display. The dialog options can be set the same as they would be set when inserting a substation from the Insert menu. This option is available only if exactly one background ellipse has been selected and no other objects have been selected.

---

## Converting Background Rectangles

---

Background rectangles can be converted to power system objects. To convert a background rectangle, right-click on a selected background line on the oneline display and the local menu will be displayed with the conversion options. The following options are available for converting a background rectangle:

### Convert to Bus

After selecting this option, the [Bus Information Dialog](#) will be displayed. The dialog options can be set the same as they would be set when inserting a bus from the Insert menu. This option is available only if exactly one background rectangle has been selected and no other objects have been selected.

### Convert to Substation

After selecting this option, the [Substation Information Dialog](#) will be display. The dialog options can be set the same as they would be set when inserting a substation from the Insert menu. This option is available only if exactly one background rectangle has been selected and no other objects have been selected.

---

## Online Fields

---

The Online Fields are fields that can be placed on an online diagram that display values specific to the current display of the diagram, such as x and y coordinate of the mouse cursor, animation rate, zoom percentage, and x and y coordinate of the center of the diagram at the center of the screen. These fields can be inserted on a online diagram by choosing **Field > Online Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.

---


## Text on Onelines

---

Text display objects are used to show single lines of text on the online.

### Edit Mode

To add descriptive text to the online, select **Background > Text** in the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab, and click the online where you would like the text to appear. The Text Options Dialog will open, asking you to enter the desired text string, and the angle at which the text is to appear on the online diagram.

To modify an existing text object, position the cursor anywhere on the text and right-click. The New Text Options Dialog will appear, allowing you to edit the text. Use the font option  of the [Formatting](#) ribbon group to control the font and background color of the text object.

---


## Memo Text

---

Memo Text display objects are used to show several lines of text on the oneline within an enclosing box.

### Edit Mode

To add a Memo Text object to the oneline, select **Background > Memo Text** from the [Individual Insert](#) ribbon group on the Draw ribbon tab. Then click and drag to create a rectangle on the oneline; this rectangle will contain the lines of text you enter.

To modify the contents of a Memo Text object, position the cursor anywhere on the object and right-click. To change the formatting of a Memo Text object, select the object with a left-click, then select  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab; this will bring up the [Font Properties](#) tab of the [Format Multiple Objects](#) dialog box.



---

## Generic Model Fields

---

The Generic Model Fields are fields that can be placed on an online diagram that display any value for any object in the case on the diagram. These fields can be inserted on an online diagram by choosing **Field > Generic Model Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.

This type of field comes in handy when you want to place a variety of fields on the diagram, without inserting specific object-type fields. All fields of data are available in one location for placement on a diagram.

---

**Pie Charts/Gauges: Lines**


---

**Pie Chart / Gauge Style**

The Pie Chart Style box determines whether the [line flow pie charts](#) show the percentage loading of the line based upon the MVA flow, the MW flow, the MVR flow, the line amp/transformer MVA flow, the maximum percentage loading under contingency, or the PTDF value. The Line Gauge Style box has the same setting options as the line flow pie charts.

**Always Use Limiting Flow**

Typically, the flow at opposite ends of a transmission line is slightly different due to losses across the line. If this option is checked, the pie chart will correspond to limiting MVA value of the line, independent of which end of the line that value occurs. Otherwise, the pie chart will always show the MVA value at the from bus for the line.

**Display Gauge Values in Percent**

Check this option to display values in the line gauges as percent of loading instead of actual flow values.

**Color, Size and Percentage**

This section of the options dialog allows you to customize the appearance of the line flow pie charts on the diagram. The default options for the line flow pie charts are set on the MVA tab, and are always in force for at least the MVA pie chart / gauge style. However, you can choose different settings for the other styles by selecting the corresponding tab under the Color, Size and Percentage section. Each of the other five styles can be set to use the same settings as defined for MVA by checking the given option on the page. If this option is unchecked for a particular style, then any pie chart on the diagram of that style type will use the specifically defined appearance options for that style type.

The following parameters are all available on each of the six style tab pages:

**Show Value Percent**

When a branch's loading exceeds the value specified in the **Show Value Percent** field, the percentage loading is shown as text within the pie chart. The default is 80%.

**Normal Size Scalar, Normal Color**

The standard, pre-warning fill color and scaling factor for pie charts.

**Warning / Limit Scalars and Colors**

The table in this section allows you to choose different settings for the pie chart size and color, based on the percentage value represented by the pie chart or gauge. Thus you can set up visual clues as to when the flow on lines exceeds specified warning or limit levels. You can add and remove points from this table by right-clicking in the table and using the Insert and Delete options from the popup menu.

To modify the Percent and size Scalar for a record in the table, simply click on the value in the cell and type in a new value. To change the color associated with the percent value, double click in the Color cell to open the Color Chooser dialog.

**Make normal color the same as the line to which the pie chart or gauge is anchored (if any)**

If checked, this option will cause the pie charts to assume the same normal colors as the transmission lines to which they are anchored. This may be useful if the transmission lines are colored according to their nominal voltage level.

**Only Apply Warning/Limit Colors and Resizing to Monitored Elements**

If this option is checked, then only those pie charts that correspond to branches selected using the [Limit Monitoring Settings](#) will change appearance to reflect warning and limit loading levels. If this option is not checked, then all pie charts will obey the options prescribed in this dialog.

Some additional options are available for pie chart appearance in situations where the line itself is out of service. Clicking on the tab labeled **Open Parameters** will display these options. You can choose to have special formatting for open devices by checking the available option, and then defining the appearance options for the pie charts on the open lines. The available options include scaling the size, width of the border, border color, pie chart background color, and drawing an "X" symbol through the pie chart.

---

**Pie Charts/Gauges: Interfaces**


---

**Pie Chart Style**

The Pie Chart Style box determines whether the interface pie charts show the percentage loading of the line based upon the MW flow, the maximum percentage loading under contingency, or the PTDF value.

**Color, Size and Percentage**

This section of the options dialog allows you to customize the appearance of the interface flow pie charts on the diagram. The default options for the interface flow pie charts are set on the Lines MVA tab. However, you can choose different settings for the interface styles by selecting the corresponding tab under the Color, Size and Percentage section. Each of the styles can be set to use the same settings as defined for Lines MVA by checking the given option on the page. If this option is unchecked for a particular style, then any interface pie chart on the diagram of that style type will use the specifically defined appearance options for that style type.

The following parameters are all available on each of the six style tab pages:

**Show Value Percent**

When an interface's loading exceeds the value specified in the **Show Value Percent** field, the percentage loading is shown as text within the pie chart. The default is 80%.

**Normal Size Scalar, Normal Color**

The standard, pre-warning fill color and scaling factor for pie charts.

**Warning / Limit Scalars and Colors**

The table in this section allows you to choose different settings for the pie chart size and color, based on the percentage value represented by the pie chart or gauge. Thus you can set up visual clues as to when the flow on interfaces exceeds specified warning or limit levels. You can add and remove points from this table by right-clicking in the table and using the Insert and Delete options from the popup menu.

To modify the Percent and size Scalar for a record in the table, simply click on the value in the cell and type in a new value. To change the color associated with the percent value, double click in the Color cell to open the Color Chooser dialog.

**Make normal color the same as the line to which the pie chart or gauge is anchored (if any)**

If checked, this option will cause the pie charts to assume the same normal colors as the interface to which they are anchored. This may be useful if the interfaces are colored differently on the diagram.

**Only Apply Warning/Limit Colors and Resizing to Monitored Elements**

If this option is checked, then only those pie charts that correspond to interfaces selected using the [Limit Monitoring Settings](#) will change appearance to reflect warning and limit loading levels. If this option is not checked, then all interface pie charts will obey the options prescribed in this dialog.

---

**Pie Charts/Gauges: Pie Chart/Gauge Styles**

---

This section shows the Pie Chart/Gauge styles in a case information display. The user can edit several of the fields directly. Otherwise, by right-clicking on any part of the case information display and selecting on **Show Dialog** in the popup menu, the [Pie Chart/Gauge Style dialog](#), where all the values can be modified.

---

## **Pie Charts/Gauges: General Options**

---

### **Show Limit Set in Header**

If this option is checked, the pie chart will display a text field showing the letter of the limit set presently being used for the element.

### **Show Style in Footer**

When checked, the pie chart will display a text field showing the style (MVA, MW, etc) that the pie chart is set to (refer to the Style property described above.)

### **Max. Zoom Percentage for Full Resizing**

The pie charts dynamically resize when you are zooming in and out on a oneline diagram. However, you can limit the point at which the pie charts resize when zooming in by setting a zoom level in this field. This helps prevent the pie charts from getting so large that they occupy the entire screen.

### **Min. Pie Chart Font Size for Warning/Limit**

Specifies a minimum font size for displaying text in the pie chart object. This is useful when zooming out on the diagram, to keep the text visible by setting a minimum font size.

### **Background Color**

The background color for the pie charts is automatically set to be the same as the online background. For a different color, select Specific color from the drop-down menu, and then click on the rectangle to the right to select the color. For a clear background, select Clear from the dropdown menu.

### **Pie Chart Relative Font Size**

Slide the slider bar to select the font size for the pie chart. This will determine the size of the font indicating the percentage on the pie chart.

### **Pie Chart Font Color**

Slide the slider bar to select the font size for the pie chart. This will determine the size of the font indicating the percentage on the pie chart.

---

## Pie Chart / Gauge Dialogs

---

There are a number of pie chart/gauge objects that can be added to one-line diagrams. These objects are inserted via the **Pies/Gauges** menu in the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab and include Areas, Buses, Generators, Injection Groups, Owners, Substations, Super Areas, Switched Shunts, Transformers, and Zones. The dialogs for these objects contain a common set of options that are required regardless of the object type. These common options as well as specific settings for the different object types are detailed below.

The [Pie Chart / Gauge Example](#) shows how the fields in this dialog affect the pie chart or gauge display. The items shown in the diagram are explained in more detail throughout this section and in the [Pie Chart / Gauge Style Dialog](#) description.

### Identification

All dialogs have a section at the top that identify the pie chart/gauge object based on the type of device that the pie chart/gauge represents. The **Find...** button is available on most dialogs to open the Find tool that can be used to advanced search for a device. Specific identifying information for the different dialogs is given below:

- **Area**The area that the pie chart/gauge represents is identified based on **Area Number** or **Area Name**. To change the area that is represented, select an **Area Number** or **Area Name** from the dropdown boxes or click the **Find** button. The dropdown boxes are populated with all areas in the case.
- **Bus**The bus that the pie chart/gauge represents is identified based on **Bus Number** or **Bus Name**. To change the bus that is represented, select a **Bus Number** or **Bus Name** from the dropdown boxes or click the **Find** button. The dropdown boxes are populated with all buses in the case with valid area/zone/owner filters.
- **Generator**The generator that the pie chart/gauge represents is identified based on **Bus Number** and **Gen ID** or **Bus Name** and **Gen ID**. To change the generator that is represented, select a **Bus Number** or **Bus Name** from the dropdown boxes and change the **Gen ID** in the edit box or click the **Find** button. The dropdown boxes are populated with all buses in the case with valid area/zone/owner filters.
- **Injection Group**The injection group that the pie chart/gauge represents is identified based on the **Injection Group Name**. To change the injection group that is represented, select a new name from the dropdown box. The dropdown box is populated with all injection groups in the case.
- **Owner**The owner that the pie chart/gauge represents is identified based on **Owner Number** or **Owner Name**. To change the owner that is represented, select an **Owner Number** or **Owner Name** from the dropdown boxes or click the **Find** button. The dropdown boxes are populated with all owners in the case.
- **Substation**The substation that the pie chart/gauge represents is identified based on **Substation Number**, **Substation Name**, or **Substation ID**. To change the substation that is represented, select a **Substation Number**, **Substation Name**, or **Substation ID** from the dropdown boxes or click the **Find** button. The dropdown boxes are populated with all substations in the case.
- **Super Area**The super area that the pie chart/gauge represents is identified based on **Super Area Name**. To change the super area that is represented, select a new name from the dropdown box or click the **Find** button. The dropdown box is populated with all super areas in the case.
- **Switched Shunt**The switched shunt that the pie chart/gauge represents is identified based on **Bus Number** and **ID** or **Bus Name** and **ID**. To change the switched shunt that is represented, select a **Bus Number** or **Bus Name** from the dropdown boxes and change the **ID** in the edit box or click the **Find** button. The dropdown boxes are populated with all switched shunts in the case with valid area/zone/owner filters.

- **Transformer**The transformer that the pie chart/gauge represents is identified based on **Near Bus Number, Far Bus Number, and Circuit** or **Near Bus Name, Far Bus Name, and Circuit**. To change the transformer that is represented, click on the Find button to use the **Find** tool.
- **Zone**The zone that the pie chart/gauge represents is identified based on **Zone Number** or **Zone Name**. To change the zone that is represented, select a **Zone Number** or **Zone Name** from the dropdown boxes or click the **Find** button. The dropdown boxes are populated with all zones in the case.

### Type of Field

Use this option to select the field to display in the pie chart/gauge. Fields that are deemed of more common interest are listed for easy access. Any field associated with a particular device can be selected by first choosing **Select a Field** and then finding the particular field in the dropdown listing all available fields or by clicking **Find Field** and searching for the field of interest.

### Current Value

#### Percent

This is the percent value calculated from the current value of the selected field. How the percent is calculated is based on the selection of limits and deadband and options associated with the Style. Details of the percent calculation are given in the **Pie Chart/Gauge Style Dialog** description.

#### Value

This is the current value of the selected field.

#### In-Service

When checked, the device represented by the pie chart/gauge is in-service. This is given for informational purposes only, as the status of devices cannot be changed from this dialog.

### Display Information

#### Type

Select to display either a Pie Chart or Gauge.

#### Gauge Orientation

If **Type** option of Gauge is selected, set this option to draw either a Vertical or Horizontal gauge.

#### Anchored

When checked, the pie chart/gauge will be anchored to the display object representing the selected device.

#### Size, Width, Use Fixed Gauge Width/Size Ratio

**Size** determines the size of a pie chart and the vertical side length of a vertical gauge or horizontal side length of a horizontal gauge. **Width** is not applicable to a pie chart but determines the horizontal side length of a vertical gauge or the vertical side length of a horizontal gauge. **Width** is not enterable if the option of **Use Fixed Gauge Width/Size Ratio** is checked. This option is checked if the **Gauge Width is Always a Fixed Proportion of Size** option is checked on the Style dialog. The fixed width/size ratio is also set on the **Style** dialog.

#### Ignore Dynamic Sizing



When checked, any sizing based on the calculated percent will be ignored and the pie chart/gauge will stay at the defined size. Options to set the sizing based on calculated percent can be found on the **Style** dialog.

#### **Ignore Dynamic Open Sizing**

When checked, any sizing of pie chart/gauges based on the device that is represented being out-of-service will be ignored and the object will stay at the defined **Size**. Options to set the sizing for open devices can be found on the **Style** dialog.

#### **Always Show Value (Percent)**

When checked, the options set on the **Style** dialog to display the percent on pie chart/gauges are ignored and the percent value is always shown.

#### **Style, Show Style Dialog**

The **Style** defines the set of generic options that can be applied to any pie chart/gauge. An already defined style can be selected from the dropdown. Click **Show Style Dialog** to modify an existing style or to create a new style; this opens up the [Pie Chart / Gauge Style Dialog](#).

### **Limits**

#### **Low Limit, High Limit**

Determines the low and high values from which the percent will be calculated and the gauge fill will be drawn. These values are not enterable if the limits are determined from the **Style** or based on device limits. More information on how these limits are used in calculating the percent is given in the **Pie Chart Parameters Tab** and **Gauge Parameters Tab** sections in the **Pie Chart/Gauge Style Dialog** description.

#### **Use Device Low Limit**

When checked, the low limit will be determined from the low limit defined for the device for the selected field type. If device limits are not defined for the selected field, this option will not be enabled. If not enabled, the low limit will be based on the value entered in the **Low Limit** box.

#### **Use Device High Limit**

When checked, the high limit will be determined from the high limit defined for the device for the selected field type. If device limits are not defined for the selected field, this option will not be enabled. If not enabled, the high limit will be based on the value entered in the **High Limit** box.

#### **Override with Limits from Style**

When checked, the high and low limits from the selected **Style** will be used.

### **Zero Percent Deadband**

#### **Use Style Zero Percent Deadband**

When checked, the zero percent high limit and zero percent low limits from the selected **Style** will be used.

#### **Zero Percent Low Limit, High Limit**

Determines the low limit and high limit for the zero percent deadband from which the percent will be calculated and the gauge fill will be drawn. These values are not enterable if the deadband is determined from the **Style**. The deadband represents the range of values for which the calculated percent will be zero. More information on how the deadband limits are used in calculating the percent is given in the **Pie Chart Parameters Tab** and **Gauge Parameters Tab** sections in the **Pie Chart/Gauge Style Dialog** description.



---

## Pie Chart / Gauge Style Dialog

---

The **Pie Chart/Gauge Style Dialog** is available from any of the pie chart/gauge dialogs by selecting **Show Style Dialog** or from **Options > Online Display Options... > Pie Charts/Gauges** on the **Pie Charts/Gauge Styles** tab.

This dialog lists options that can be commonly applied to pie chart/gauge objects so that these same options do not have to be set for every individual object. The defined styles can be applied to any of the new pie chart/ gauge objects that include Areas, Buses, Generators, Injection Groups, Owners, Substations, Super Areas, Switched Shunts, Transformers, and Zones. These styles are currently not applicable to Lines and Interfaces. Some options are related to options that are set for individual pie chart/gauges. More information about individual object settings can be found in the **Pie Chart/Gauge Dialogs** section.

The [Pie Chart / Gauge Example](#) shows how the fields in this dialog affect the pie chart or gauge display. The items shown in the diagram are explained in more detail throughout this section and in the [Pie Chart / Gauge Dialogs](#) description.

### Creating and Saving Styles

#### Style Name

Drop-down box lists all currently defined pie chart/gauge styles. Select **Add New** to create a new style. Select **Rename** to change the name of an existing style. Select **Delete** to remove an existing style.

#### Save AXD

Save all currently defined pie chart/gauge styles to a Display Auxiliary file.

#### Load AXD

Load pie chart/gauge styles from a Display Auxiliary file.

### Hide all Style Objects (except in edit mode)

When checked, the pie chart/gauge will not be visible in run mode.

### Show Header (Object ID)

When checked, the header of the pie chart/gauge will contain identifying information about the device that the pie chart/gauge represents.

### Show Footer (Field Type)

When checked, the field type that is being used in the pie chart/gauge will be displayed in the footer of the pie chart/gauge.

### Default Style

When checked, this style is one of the default styles that are defined. Default styles cannot be renamed or deleted. This is an informational field and cannot be changed by the user.

### Limits and Zero Percent Deadband Values

Define the High and Low Limits to use when the option **Override with Limits from Style** is selected for the pie chart/gauge that is using this style and the High and Low Zero Percent Deadbands to use when the option **Use Style Zero Percent Deadband** is selected for the pie chart/gauge that is using this style. The deadband represents the range of values for which the calculated percent will be zero. More information is given in the [Pie Chart Parameters Tab](#) and [Gauge Parameters Tab](#) sections on how the limits are used in calculating the percent.

The remainder of the dialog is broken down into four tabs:

- [Standard Parameters](#)

## Building a Oneline Diagram

- [Open Parameters](#)
- [Pie Chart Parameters](#)
- [Gauge Parameters](#)

---

**Pie Chart / Gauge Style Dialog - Standard Parameters Tab**


---

**Normal Scalar, Normal Fill Color**

These are the standard, pre-warning scaling factor and fill color for pie chart/gauges. To change the color, click on the color to open the Color dialog.

**Font Color**

Font color of the text displayed in the pie chart/gauge. To change the color, click on the color to open the Color dialog.

**Border Width**

Width of any border portion of the pie chart/gauge. The border of a pie chart encloses the entire pie chart as well as the filled portion of the pie chart. The border of a gauge encloses the entire gauge. The border width also impacts the width of the lines marking the limits and deadband on the gauge.

**Border Color**

Color of any border portion of the pie chart/gauge. The border of a pie chart encloses the entire pie chart as well as the filled portion of the pie chart. The border of a gauge encloses the entire gauge. The border color also is the color used for the lines marking the limits and deadband on the gauge. To change the color, click on the color to open the Color dialog.

**Border Color Same as Fill Color**

When checked, the border color will be the same color as the fill color and the color selected in the Border Color will be ignored.

**Background Color**

The drop-down box allows selection of three options for the background color: Oneline Background, Specific Color, and Clear. Oneline Background will color the pie chart/gauge background the same color as the selected background color for the oneline. Specific Color will allow the selection of a user specified color. To specify a color, click the box to the right of the drop-down to open the Color dialog. The Clear option will not use a background color and objects underneath the pie/chart gauge will be visible through the unfilled portions of the pie chart/gauge.

**Relative Font Size**

Modify the track bar to adjust the size of the font relative to the size of the pie chart/gauge.

**Max Zoom Percent for Full Resize**

Pie chart/gauges dynamically resize when zooming in and out on oneline diagrams. A limit can be placed on the point at which the pie chart/gauges resize when zooming in by setting a zoom level in this field. This helps prevent the pie chart/gauges from getting so large that they occupy the entire screen.

**Warning/Limit Scalars and Colors**

The table in this section allows the definition of different settings for pie chart/gauge size and fill color based on the percent value represented by the pie chart/gauge. Points can be added or removed from this table by right-clicking in the table and using the Insert and Delete options from the popup menu.

To modify the Percent and size Scalar for a record in the table, click on the value in the cell and type in a new value. To change the color associated with the percent value, double click in the Color cell to open the Color dialog.

**Use Discrete Map**

When checked, discrete colors will be used for the pie chart/gauges based on the colors specified in the table. Otherwise, colors will be blended if the percentage represented in the pie chart/gauge falls between percentages defined in the table.

---

**Pie Chart / Gauge Style Dialog - Open Parameters Tab**

---

Open parameters are applied to any pie chart/gauges when the device that they represent is out-of-service.

**Special Formatting for Open Devices**

When checked, the **Open Size Scalar**, **Open Border Width**, and **Open Border Color** options are applied when a device is out-of-service.

**Use Special Open Background Color**

When checked, the **Open Background Color** option is applied.

**Open Symbol**

When an **Open Symbol** other than None is selected, the symbol will be drawn in the selected **Open Symbol Color**.

---

**Pie Chart / Gauge Style Dialog - Pie Chart Parameters Tab**


---

Options given on this tab only apply to how pie charts are displayed.

**Show Value Percent**

When the percent of the pie chart/gauge exceeds the value specified in this field, the percent is shown as text within the pie chart/gauge. The default is 80%.

**Display of Values Below the Zero Percent Deadband**

The selection of this option determines how to calculate the percent represented in pie charts. Four parameters are required for determining this: low limit, high limit, zero percent low limit, and zero percent high limit. These four limits are determined from options set with individual pie charts and/or options set with the style. In general, the zero percent low limit and zero percent high limit define the deadband in which the percent is zero. The percent is then determined based on where the selected field value falls within the high limit and zero percent high limit and the low limit and zero percent low limit. The specific calculations used for determining the percent is given for the options below. In all cases if the limits are not defined correctly (i.e., none of the If..then statements below apply), the percent will be returned as zero.

- **Allow negative percent values**
  - o If (field value) > (zero percent high limit) and (high limit) > (zero percent high limit) then  

$$\text{Percent} = ((\text{field value}) - (\text{zero percent high limit})) / ((\text{high limit}) - (\text{zero percent high limit}))$$
  - o If (field value) < (zero percent low limit) and (low limit) < (zero percent low limit) then  

$$\text{Percent} = ((\text{field value}) - (\text{zero percent low limit})) / ((\text{zero percent low limit}) - (\text{low limit}))$$
- **Use absolute value** (i.e., percent always >= 0)
  - o Same calculations as in **Allow negative percent values** except that the absolute value of the calculated percentage is reported.
- **Low limit is zero percent**
  - o The zero percent deadband limits are ignored in this option and the percent is calculated based on the low and high limits only
  - o If (high limit) > (low limit) then  

$$\text{Percent} = ((\text{field value}) - (\text{low limit})) / ((\text{high limit}) - (\text{low limit}))$$
- **Treat values below zero deadband as zero**
  - o If (field value) > (zero percent high limit) and (high limit) > (zero percent high limit) then  

$$\text{Percent} = ((\text{field value}) - (\text{zero percent high limit})) / ((\text{high limit}) - (\text{zero percent high limit}))$$
  - o If (field value) < (zero percent low limit) and (low limit) < (zero percent low limit) then  

$$\text{Percent} = 0$$



---

**Pie Chart / Gauge Style Dialog - Gauge Parameters Tab**


---

Options given on this tab only apply to how gauges are displayed.

**Show Limit Labels**

When checked, text labels will be drawn with the gauge showing the four limits in use.

**Include Units in Limit Labels**

When checked, units will be included when the limit labels are shown.

**Numeric Value Type**

This option allows the numeric value to either be displayed as the **Actual Value** or the calculated **Percent** value.

**Display of Numeric Value**

This option determines when the value of the field is displayed with the gauge. **Never** will not show the value, **Always** will always show the value, and **If Violating Warning/Limit Value** will only show the value if it is outside of the limit range that is currently in use.

**Digits to Right of Decimal**

This parameter determines how many digits to the right of the decimal are displayed in the limit labels and the numeric value when they are shown with the gauge.

**Gauge Width is Always a Fixed Proportion of Size**

When checked, the **Gauge Fixed Width/Size Ratio** is used to determine the width of the gauge as a proportion of the size (height). Note: the size of gauges is specified for individual gauges on the pie chart/gauge dialogs.

**Gauge Fixed Width/Size Ratio**

Specifies the ratio of the gauge width to the size when the **Gauge Width is Always a Fixed Proportion of Size** option is checked.

**Percent Overhang on Top of Gauge**

The overhang at the top of the gauge is the region above the high limit line. The size of this region is defined as a percentage of the total size of the gauge.

**Percent Overhang on Bottom of Gauge**

The overhang at the bottom of the gauge is the region below the low limit line. The size of this region is defined as a percentage of the total size of the gauge.

**Gauge Fill and Percent Calculation**

These options determine how the calculated percent will be displayed and how the gauge will be filled. Four parameters are required for determining these: low limit, high limit, zero percent low limit, and zero percent high limit. These four limits are determined from options set with individual gauges and/or options set with the style. In all cases if the limits are not defined correctly, the percent will be returned as zero.

**Use Absolute Value for Percent**

When checked, the displayed percent will be the absolute value of the calculated percent. This option does not impact how the gauge will be filled.

**Gauge Fill Options (Zero Percent Reference)**

This option determines how the gauge will be filled. If **Fill up or down from deadband** is selected, the fill will begin at the low or high deadband limit and continue up or down depending upon where the field value falls within the defined limits. If **Fill**

**up from bottom** is selected, the fill will begin at the bottom of the gauge and fill up to the field value.

Regardless of which fill option is selected, the following equations will be used to calculate the percent based upon where the field value falls within the defined limits:

If  $(\text{field value}) > (\text{zero percent high limit})$  and  $(\text{high limit}) > (\text{zero percent high limit})$  then

$$\text{Percent} = ((\text{field value}) - (\text{zero percent high limit})) / ((\text{high limit}) - (\text{zero percent high limit}))$$

If  $(\text{field value}) < (\text{zero percent low limit})$  and  $(\text{low limit}) < (\text{zero percent low limit})$  then

$$\text{Percent} = ((\text{field value}) - (\text{zero percent low limit})) / ((\text{zero percent low limit}) - (\text{low limit}))$$

### **Ignore Deadband**

When checked, deadband limits are ignored. This effectively forces the fill to start from the bottom of the gauge and fill up to the field value. The percent is calculated based on the following equation and will always be reported as the absolute value:

If  $((\text{high limit}) - (\text{low limit})) < > 0$  then

$$\text{Percent} = ((\text{field value}) - (\text{low limit})) / ((\text{high limit}) - (\text{low limit}))$$

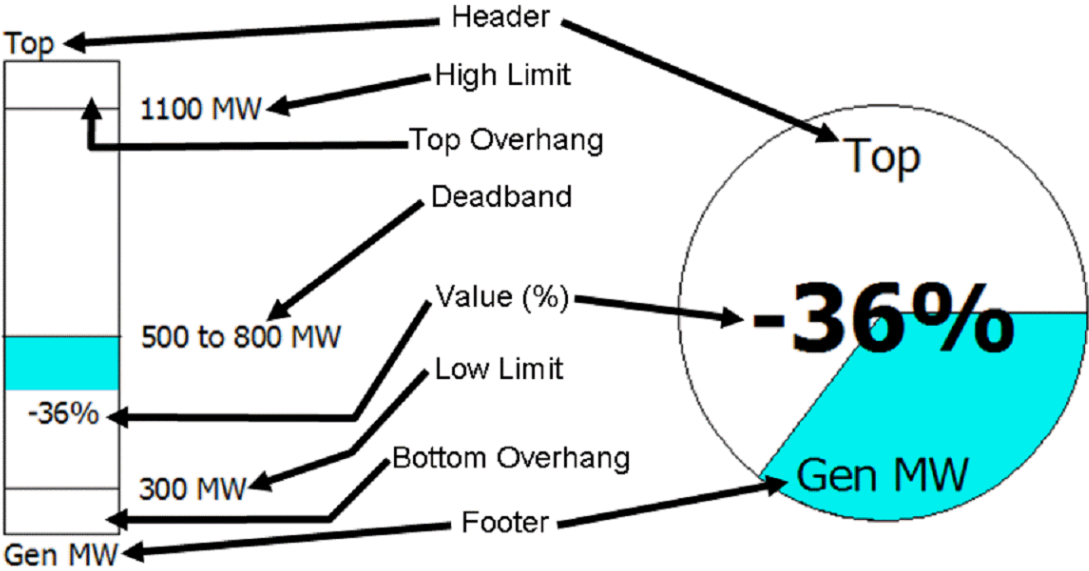
### **Override Low Limits with a Value of Zero**

When checked, the low limit value will be set to zero when determining the fill and calculating the percent. This option is only applied if the **Ignore Deadband** check box is also checked. The percent is calculated based on the following equation and will always be reported as the absolute value:

If  $((\text{field value}) > 0)$  and  $((\text{high limit}) < > 0)$  then

$$\text{Percent} = (\text{field value}) / (\text{high limit})$$

Pie Chart / Gauge Example



---

## Chapter 5: Edit Mode Tools and Options

---

Edit Mode provides a number of commands for customizing and modifying the case that are different from Run Mode.

This chapter covers the following:

- General Edit Mode Tools
- Insertion Using Palettes
- Automatic Object Insertion
- Equivalencing
- System Scaling
- Facility Analysis

---

## Appending a Case

---

The Append Case command allows you to append additional power system components to an existing case. Unlike the [Open Case](#) command, Append Case does not delete the existing case (if any) before loading the selected case. To append a case to the existing case, go to the [Tools](#) ribbon tab and choose **Modify Case > Append Case** from the [Edit Mode](#) ribbon group.

The Append Case command can be useful when used in conjunction with the [Equivalencing Display](#).

### Notes

Appended data completely overwrites an existing record corresponding to the same bus number(s).

Branch elements are appended to a case only if both of their terminal buses exist.

---

## Refresh Anchors

---

The **Refresh Anchors** option of the [Formatting](#) ribbon group on the [Draw](#) ribbon tab allows you to update or reset the anchoring of objects on a oneline diagram to their respective anchor. It can be advantageous to use the Refresh Anchors option when opening onelines that were created by another case, copying and pasting data between onelines, or renumbering objects on the oneline. This ensures the anchoring of objects to the appropriate anchor on the oneline diagram.

---

## Bus Renumbering Dialog

---

The Renumber Buses Display allows you to change the bus numbers for either the entire power system case, and/or for any open oneline diagrams. To show this display, while in [Edit Mode](#), go to the [Tools](#) ribbon tab, and choose **Renumber > Renumber Buses** from the [Edit Mode](#) ribbon group.

The bus renumbering feature is provided because it is sometimes necessary to renumber buses in the power system model, either to make room for new buses or to move buses to a different zone or area that has a different numbering scheme. It may also be necessary to renumber bus display objects on the oneline if you want to use the oneline with a case other than the one for which it was originally designed. The bus renumbering feature provides a convenient way of accomplishing this.

The table at the bottom of the Bus Renumbering Dialog is used to manage the lists of current bus numbers and any desired changes to the numbering scheme. You can specify the bus numbers to change and their new values by directly typing them into the table. Alternatively, you can generate the bus list automatically by selecting one of the [Automatic Setup of Bus List Options](#) and clicking the **Setup Bus Swap List** button. The Automatic Setup options allow you to add to the list all buses in the case, all buses in the case subject to the [area/zone/owner filter](#) constraints, all buses currently displayed on the oneline, or a set of numbers from a text file.

The table behaves just like a [Case Information Display](#) and thus has a [local menu](#) that can be invoked by clicking the right mouse button. Select **Insert** from the local menu to insert a new bus number to change, or select **Delete** to delete a bus number that is currently in the list. Select **Show Dialog** to display the [Bus Information Dialog](#) corresponding to the bus number on which you right-clicked. You may clear the entire list by pressing the **Clear Bus List** button.

Once you have indicated which buses you would like to renumber in the table, select an option from **Bus Change Options** to specify where you would like to implement the changes (in both the case and the oneline, in the case only, or in the oneline only). Make the changes by pressing the **Change Bus Numbers** button. Close the dialog by pressing **Close**.

The Automatic Setup of Bus List and Bus Change options deserve further discussion. See for further details or click [NEXT](#).

---

## Bus Renumbering: Automatic Setup of Bus List Options

---

This section of the [Bus Renumbering Dialog](#) is used to generate automatically a list of the buses to be renumbered. Select an option from the list and click on the **Setup Bus Swap List** button to generate the list, or **Clear Bus Swap List** to clear the list.

### Load All Buses in Cases

Creates entries in the table for every bus in the system. By default the new bus number is the same as the old bus number. Of course you do not have to renumber every bus. If you would like a bus to keep its same number, either add that bus to the table, or simply have an entry with the old and new bus numbers identical.

### Load Buses with Valid Area/Zone Filters

Same as the **Load All Buses In Case** option except only those buses with valid [area/zone/owner filters](#) are added. This allows you to easily modify the bus numbers in just an area or zone.

### Load Only Buses on Oneline

Creates entries in the table only for those buses on the current oneline. This option is most useful when you are just changing the buses on the onelines.

### Load Buses From Text File

Creates entries in the table using an external text file. The format of this file is as follows:

```
1 11 Specifies converting from old bus number 1 to new bus number 11
2 22 Specifies converting from old bus number 2 to new bus number 22
etc.
```

### Freshen Current Oneline

The Freshen Current Oneline option is designed to help you quickly renumber an existing oneline to work with a new numbering scheme. You will find this method helpful if you have been using the oneline with a case and now must use it with a different case having a different set of bus numbers, but the same bus names. Freshen Current Oneline will try to match the buses on the oneline with the buses in the new case by matching bus names and kV, rather than by number (which is how Simulator usually tries to link bus display objects with bus records in the case). The best way to learn how to use Freshen Current Oneline is to consider the following example:

To update an old oneline to work with a new bus numbering scheme:

- Open the oneline and the old case with which it was used.
- Choose **Renumber > Renumber Buses** from the [Edit Mode](#) ribbon group on the [Tools](#) ribbon tab.
- Select **Load Only Buses on Oneline** and press the **Setup Bus Swap List** button.
- Change the **Swap?** field values all to *Yes*.
- Right click on the table and choose **Save List to File**. Give the file a name. For this example, we'll name the file "oldscheme.txt." This file will contain the list of buses represented on the oneline, specifying each bus's number, name, kV, and area.
- Close the old case.
- Open the new case and the oneline you wish to renumber. If any other onelines open with the case, close them. You want only the oneline you wish to renumber to be shown.



- Choose **Renumber** > **Renumber Buses** from the [Edit Mode](#) ribbon group on the [Tools](#) ribbon tab.
- Select **Freshen Current Online** and specify the file "oldscheme.txt" (for this example). Click the **Setup Bus Swap List** button. Simulator will match the old numbering scheme used in the online with elements in the new case by name and kV. If it finds more than one match, it will use the element's area name as a tie breaker. If it still can't reconcile the multiple matches, it will add both renumbering options to the table.
- Go through the new list and make sure that you want to swap the buses that are listed. If you do, change the **Swap?** field value for each to *Yes* (you can do this quickly for all buses by right-clicking on the **Swap?** column and choosing **Toggle All Yes**). Be sure to reconcile any duplicate bus renumbering suggestions. These are cases for which Simulator could not determine how to renumber the buses because a bus on the diagram matches more than one bus in the case by name, kV, and area.
- Click the **Change Bus Numbers** button at the bottom of the form.

Once the online has been renumbered, save it with the case by selecting **Save Case** (or **Save Case As** if you wish to give it a different name) from the [Application Button](#). See [Bus Change Options](#) for additional details.

---

**Bus Renumbering: Bus Change Options**

---

This option is used to specify which buses to change.

**Change Both Case and Onelines (default)**

Renumbers the buses in both the case and any open oneline diagrams.

**Change Only Case**

Only renumbers the buses in the case. The oneline bus numbers are not changed.

**Change Only Onelines**

Only renumbers the buses on the onelines. The case itself is not changed. You would want to select this option if you have already changed the case (or loaded a different one), but now have several onelines based on that case that also need to be changed. See [Automatic Setup of Bus List Options](#) for more details. This is the most commonly used option.

---

## Renumber Areas/Zones/Substations Dialog

---

The Renumber Areas/Zones/Substations Dialog is available while in [Edit Mode](#), by going to the [Tools](#) ribbon tab, and choosing **Renumber > Renumber Areas/Zones/Substations** from the [Edit Mode](#) ribbon group. This dialog allows the renumbering of areas, zones, and substations for either the entire power system, and/or for any open oneline diagrams. This dialog is similar to the [Renumber Buses Dialog](#).

This feature is provided because it is sometimes necessary to renumber areas, zones, or substations in the power system model, either to make room for new elements or to move elements to different area, zones, or substations that have a different numbering scheme. It may also be necessary to renumber display objects on the oneline if the oneline is used with a case other than the one for which it was originally designed. The areas/zones/substations renumbering feature provides a convenient way of accomplishing this.

The table at the bottom of the Renumber Areas/Zones/Substations Dialog is used to manage the list of areas, zones, and substations available for renumbering. This list is populated by selecting one of the **Automatic Setup of Swap List** options. The Automatic Setup options allow the addition of all areas, zones, or substations in the case or only those on the oneline. An option is also available to read the swap information from a file or to setup the swap list based on the current oneline. The **Types to Insert** option is available for specifying whether or not to add areas, zones, and substations to the swap list. Once the swap list option and types to insert have been selected, click the **Setup Swap List** to populate the table.

The table behaves just like a [Case Information Display](#) and thus has a [local menu](#) that can be invoked by clicking the right mouse button. Select one of the **Insert** options from the local menu to insert a new area, zone, or substation number to change, or select **Delete** to delete an element number that is currently in the list. The entire list may be cleared by clicking the **Clear Swap List** button. The **Present Number** for the element is specified in the list. Change the **New Number** field to specify the updated number of the area, zone, or substation.

The **Renumber Options** allows the specification of where the renumbering changes should be implemented. The changes can be applied to the power system case only, the oneline only, or both the case and the oneline.

Once the elements to renumber have been specified and the **New Number** fields have been set appropriately, implement the changes by pressing the **Renumber** button. The changes will be applied to the case and/or oneline as specified in the **Renumber Options**. Close the dialog by pressing **Close**.

---

## Merging Buses

---

Two or more buses can be merged to a new bus in Edit Mode with the loads, generators, and shunts of the buses merged moved to the new bus. The transmission lines among the buses merged will be deleted while the transmission lines connecting the merged buses and buses which are not selected to merge will be moved to the new bus.

To join two or more buses together using the oneline display, select at least two power system elements (at least one of them being a bus), then right click on one of the selected elements to invoke the popup menu and select **Merge selected buses**. In the Bus Merging Dialog, specify the buses to merge further if needed by clicking **Add** and/or **Delete** button. To add buses to merge, click **Add** and select all the needed buses in the **Choose a Bus** dialog and click **OK**. To delete from already selected buses, select the buses to delete and click the **Delete** button.

Once you have the elements to merge selected, enter the number, name, nominal voltage, zone number, area number and substation number for the new bus. These properties can be set to be the same as one of buses to merge by selecting it in the **Specify buses to merge** box and pressing the **Set new bus properties same as selected bus**.

Buses can also be merged in the [Bus Case Information Display](#). To do so, select a cell and right click to popup the local menu. Select **Merge selected buses**. The buses to merge and the properties of the new bus can be edited in the Bus Merging Dialog.

When buses are merged from the oneline diagram, the selected buses will be joined in both the PWB case and the oneline. When buses are merged from the bus grid, the selected buses will be joined only in the PWB case but not in the oneline. This might result in bus objects not connected to bus records in the oneline.

---

## Splitting Buses

---

Simulator assists you in transforming one bus into two connected buses. This process is called splitting a bus. In performing the split, the user is able to decide which equipment to keep connected to the original bus and which equipment to move to the new bus. Because this activity impacts the structure of the power flow equations, bus splitting can be performed only in [Edit Mode](#).

To split a bus from the oneline diagram, right click on the bus you wish to split, and click **Split Bus** from the resulting local menu. To split a bus from the [Bus Case Information Display](#), right-click on its corresponding record and again click **Split Bus** from the resulting local menu. In either case, the [Split Bus Dialog](#) will appear.

[Multi-section lines](#) merit special consideration during bus split operations. These are the rules Simulator follows when you try to split a bus that is part of a multi-section line. If the original bus is the endpoint bus of a multi-section line and the ending line segment was transferred to the new bus, then the new bus becomes the new ending terminal of the multi-section line. If the original bus was a dummy bus of the multi-section line, and if exactly one of the branches connected to the original bus is rerouted to the new bus, then both the new bus and the original bus will be dummy buses in the reconstituted multi-section line. If neither or both of the lines connected to the original dummy bus were rerouted to the new bus, then the multi-section line definition is eliminated, since Simulator has no way to determine how the multi-section line should be redefined.

A final consideration involving bus splits is how [sequence data](#) is treated. If you have defined sequence data for fault analysis, Simulator will recalculate the sequence data for the original and new buses after the split. In this case, the zero sequence impedance for the new branch that connects the original and the new bus will be set to j0.0001.

---

## Split Bus Dialog

---

The Split Bus dialog contains two sections labeled **Existing Bus** and **New Bus**. These sections enable you to designate the bus you want to split and the name and number of the new bus created by the bus split operation. If you opened the Split Bus Dialog from either the [Bus Case Information Display](#) or an oneline diagram, the Existing Bus Name and Number fields will be read-only and will identify the bus you selected. However, if you opened the Split Bus Dialog from the **Modify Case Menu** on the [Edit Mode](#) ribbon group on the [Tools](#) ribbon tab, you will first have to choose a bus to split. In this case, use the **Find...** speed button located next to the **Existing Bus** label to select the bus to split, or simply type its number in the Existing Bus Number field. Then, provide the name and number for the new bus to create using the New Bus Name and Name fields. The number you specify in the New Bus Number field must be unique; it cannot be a number that identifies another bus in the case. If you do specify an existing number, Simulator will issue an error message and require you to specify a different number. If you choose not to specify a name for the new bus, Simulator will set the name of the new bus to be the same as the bus's number.

Once you have identified the bus to split and the name and number for the new bus resulting from the split, you may then specify whether a bus tie should be inserted between the existing bus and the soon-to-be-created new bus. By checking the **Insert bus tie between existing and new buses** checkbox, you command Simulator to place a very low-impedance bus tie between the bus to split and its offspring. The new branch will have an impedance of  $0.0000 + j0.0001$  ohms. If the bus tie should be inserted as an open branch, check the **Normally open** checkbox. This option becomes available, of course, only if you elect to have Simulator automatically add the new bus tie.

After you have finished making your selections, click **OK**. Simulator will create the new bus, assign its electrical attributes to match those of the existing bus, and add the bus to the power system model. If a oneline diagram is currently active and the existing bus is represented on it, Simulator will add a symbol for the new bus to the diagram, placing it immediately to the right of the existing bus's symbol. If you elected to create a bus tie between the existing and new buses, Simulator will also add a symbol for the bus tie to the diagram.

Finally, Simulator will automatically open the [Equipment Mover Dialog](#) to help you manage the transfer of equipment from the existing bus to the new bus.

---

## Equipment Mover

---

Simulator provides a convenient tool for transferring equipment between buses. Simulator allows you to move bus shunts, loads, generators, switched shunts, and transmission lines between buses. For loads, generators, and both varieties of shunts, Simulator offers you the ability to transfer all or part of the equipment from the origin bus to the destination bus.

Equipment may be transferred between buses using the Equipment Mover Dialog. The Equipment Mover Dialog can be opened in any of four ways:

- Go to the [Tools](#) ribbon tab and choose **Modify Case < Move Bus Equipment...** from the [Edit Mode](#) ribbon group.
- On the oneline display right-click on the bus from which you want to move equipment and select **Move Equipment...** from the resulting local menu.
- From a [Bus Case Information Display](#), right-click on the record corresponding to the bus from which you want to move equipment and select **Move Equipment ...** from the resulting local menu.
- As the final step of the [Split Bus](#) operation.

Regardless of the approach you take, Simulator will then open the Equipment Mover Dialog. The Equipment Mover Dialog consists of three sections. The top portion of the dialog is split in two sections that identify the bus from which equipment will be transferred (on the left) and the bus equipment will be transferred to (on the right). If the dialog was opened as the final operation of the [Split Bus](#) operation, these two buses will be hard-coded to identify the original bus and the bus resulting from the split. If the dialog was opened from the main menu, you must select both the origin and destination buses from lists that are reminiscent of [case information displays](#). If the dialog was opened using one of the other two methods, the origin bus will be set to the bus you selected, and you will then have to choose the destination bus. To select a bus to be an origin or destination bus, simply select the corresponding record from the appropriate list.

Once the origin and destination buses have been identified, you must then select the equipment to transfer from the origin to the destination bus using the case information display that occupies the bottom of the dialog. To move a particular piece of equipment, toggle the value of the **Move Object?** field to **YES**. To move just portions of loads, generators, or shunts from the origin to the destination bus, adjust the value of the **Move %** field from 100.0 to the percentage you desire.

Once you have selected the equipment you wish to transfer, click the **Move equipment** button. Simulator will adjust the power system model to reflect your equipment transfer requests. Furthermore, Simulator will provide you an opportunity to manually adjust all open oneline diagrams to reflect the equipment transfers. To facilitate this activity, Simulator will open a [Potential Misplacements Dialog](#) for each oneline that displays the origin bus. The Potential Misplacements Dialog lists the display objects associated with the equipment that had just been transferred from the origin to the destination bus. By clicking on an entry in this list, you can pan the associated oneline diagram to focus on that object. This allows you to identify display objects that perhaps should be relocated to reflect their new bus associations. Once you have finished addressing these potentially misplaced display objects, click the **OK** button to close the Potential Misplacements Dialog.

If you find that you would like to reopen the [Potential Misplacements Dialog](#) after you have closed it, click the **List most recent transfers** button. This will reopen the Potential Misplacements Dialog associated with the most recent equipment transfer operation.

To conclude the equipment transfer operation, click the **Close** button.

---

## Potential Misplacements Dialog

---

In the wake of an equipment transfer operation using the [Equipment Mover Dialog](#), some oneline display objects may be out of place. To ease the task of correcting these misplacements, Simulator provides the Potential Misplacements Dialog. Simulator opens a Potential Misplacements Dialog for each oneline that displays the bus that served as the origin for equipment transfer. The Potential Misplacements Dialog lists the display objects associated with the equipment that had just been transferred from the origin to the destination bus. By clicking on an entry in this list, you can pan the associated oneline diagram to focus on that object. This allows you to identify display objects that perhaps should be relocated to reflect their new bus associations. Once you have finished addressing these potentially misplaced display objects, click the **OK** button to close the Potential Misplacements Dialog.



---

## Tapping Transmission Lines

---

Simulator eases the process of inserting a bus at some location along an existing transmission line. This feature can be extremely useful when you want to add a new generation site to a model, for example. Rather than having to delete an existing line, place the bus, and draw two new transmission lines, Simulator simplifies the task to a one-step process.

A line can be tapped from a oneline diagram, the transmission line case information display, or by going to the [Tools](#) ribbon tab and choosing **Modify Case > Tap Transmission Line** from the [Edit Mode](#) ribbon group. From the oneline diagram, right-click on a transmission line and select **Insert Line Tap** from the popup menu. From the [case information display](#), simply right-click on the corresponding branch record and select **Tap Transmission Line** from the popup menu. Any of these methods will open the [Automatic Line Tap Dialog](#) for setting up and inserting the new bus.

Note that transmission lines may be split only from [Edit Mode](#). You cannot access this functionality from [Run Mode](#).

---

## Automatic Line Tap Dialog

---

The Automatic Line Tap Dialog will allow you to define the settings to use for inserting a new bus along a transmission line. The transmission line to tap will be represented in the panel at the top of the dialog, displayed as a **Near Bus** and **Far Bus**. If the Automatic Line Tap Dialog was opened by right-clicking a line on a oneline diagram, or by right-clicking on a record in the case information display, the line will already be selected in the panel. You can change the line selection by first choosing the **Near Bus** you desire, and then selecting the **Far Bus** from the list of possible connections to the chosen **Near Bus**. Note that the percentage entered in the **Position along line** field will be in relation to the selected **Near Bus**. The inserted bus and new sections of the tapped line will adhere to the following settings:

### Position along line

The field labeled **Position along line** will indicate the point where you right-clicked the mouse relative to the location of the Near Bus to the mouse-click in terms of a percentage of the total line length. In other words, if the line is 10 units long, and you clicked the right mouse at location 7 units from the Near Bus end of the line, the **Position along line** will indicate 70%. If you opened the Automatic Line Tap Dialog from the case information display or the **Tools** ribbon tab, the **Position along Line** will be set to 50%. In either case, you can adjust this setting to place the new bus more precisely. The placement of the new bus controls how the impedances of the new lines are set, as the impedance of each section will equal the section's corresponding percentage length multiplied by the impedance of the original line. Note that the original charging capacitance of the line will be reassigned as determined by the selection under the **Shunt Model** option.

### New Bus Number

By default, Simulator will find and set an unused bus number for you, but you can specify the number to be used for the new bus, between 1 and 2147483647. If you enter a bus number that already exists, you will be prompted to enter a different number when you click the **Tap** button.

### New Bus Name

Specify a name to be assigned to the new bus. By default this field is blank, and if left blank Simulator will set the name of the bus the same as the new bus number.

### New Bus Area

You can specify the area for the new bus to be the same as the **Near Bus**, the **Far Bus**, or another value of your specification. If you select the **Specify** option, the edit box and find button will become enabled. You can then enter an area number manually, or click **Find** to locate an area from the list of areas currently in the case. If you want a new area to be assigned to the case for this bus, simply enter an unused area number manually in the box, and Simulator will automatically set up the new area record for the case. You can then open the [Area Information Display](#) and set the name and other values for the new area.

### New Bus Zone

You can specify the zone for the new bus to be the same as the **Near Bus**, the **Far Bus**, or another zone of your specification. If you select the **Specify** option, the edit box and find button will become enabled. You can then enter a zone number manually, or click **Find** to locate a zone from the list of zones currently in the case. If you want a new zone to be assigned to the case for this bus, simply enter an unused zone number manually in the box, and Simulator will automatically set up the new zone record for the case. You can then open the [Zone Information Display](#) and set the name and other values for the new Zone.

### Shunt Model

By default, Simulator will reassign the original charging capacitance as line shunts at the original terminal bus ends of the two new line segments. The charging capacitance of the two new branch elements will be set to 0. This will result in an exact match of power flows before and after the line tap.

Alternatively, you can choose to have the capacitance converted approximately using percentage entered as the **Position along line**. In other words, the original capacitance will be scaled and assigned to the two new segments as charging capacitance. This will result in a slightly different power flow result after the line tap, as it is an approximation of charging capacitance on the two new segments.

### Treat sections as a multi-section line

Check the box labeled **Treat sections as a multi-section line** to force the status of the two new line sections to be controlled in unison. Checking this box will cause a multi-section line to be created comprised of the two new line sections. See [Multi-Section Line Information](#) for details.

Click the **Tap** button to close the Automatic Line Tap Dialog and perform the line tap. If a value on the dialog is not set properly, a warning message will appear, and you will need to either change the specified value or cancel the process.

---

## Setting Default Drawing Options

---

The Default Drawing Options Dialog is used to define the default sizes of new display objects, as well as various other display parameters. These options are available only in Edit Mode.

To open this dialog choose **Default Drawing Values** on the **General Options** ribbon group of the [Options](#) ribbon tab. These General Options are also available on the [Online](#) ribbon tab.

The Default Drawing Values Dialog is organized using a list showing the types of objects that can be added to onelines. To view information about a particular type of object, select that object type from the list and tables which describe the defaults available for that object will be populated. Two tabs will be available. One will contain the majority of default options that are available for the selected display object type, and the other will contain the default positions of fields that will be inserted when the object is inserted.

Generally, the Default Drawing Values apply when you insert *new* objects on an oneline diagram. Changing the Default Drawing Values does not automatically affect the *existing* objects already drawn on the diagram. However, you can select multiple objects on a oneline diagram, and right-click on any one object to invoke the object's popup menu. You will find the option **Apply Default Draw Values**, which will modify the formatting of the selected objects to meet the default drawing value specifications.

### Variable Defaults depending on Voltage Level

Many kinds of objects can have more than one set of defaults specified depending on their voltage level. When more than one set of defaults is specified, the table is always sorted by Nom kV.

A new object is inserted with properties corresponding to the default with the lowest Nom kV that the object's nominal kV is larger than. If the object's nominal kV is smaller than all defaults, then it will be set according to the default with the lowest nominal kV.

For instance, assume 4 sets of defaults are defined for buses roughly as follows

Nom kV Size

>400 15

>300 10

>200 8

>100 5

A new bus with nominal voltage of 345 kV is inserted with size 10.

A new bus with nominal voltage of 299 kV is inserted with size 8.

A new bus with nominal voltage of 69 kV is inserted with size 5.

In order to add another set of defaults for a kind of object, right-click on the table and choose **Insert** from the local menu. In order to delete a set of defaults, right-click on the table and choose **Delete**. Once you have inserted a new default, specify the new Nom kV for the default along with the new defaults.

Notes: When you change the Nom kV of a default, the list always sorts itself according to Nom kV. **Insert** will not be enabled in the local menu for objects that cannot have more than one set of defaults. **Delete** will not be enabled if only one set of defaults is defined.

### Modifying Values in the Default Drawing Values Dialog

The font colors in tables of this dialog follow the conventions of the [Case Information Displays](#). Most default values are enterable and can be modified by selecting the value you want to change and then typing in the new value. Other values are toggleable and can be changed by double-clicking on the value. For reference regarding what the various "size" defaults mean, when the zoom level is at its nominal value (100%), the size of the screen is 100 by 100. There are also some special kinds of fields in this dialog which are discussed next.

### Editing Colors and KV-Based Color Coding

In order to modify a color value, double-click on the colored rectangle to bring up the Color Dialog. Then specify the color you want. For some defaults, such as fill color, it is possible to specify the default color to be *none*. In order to set a color default to *none*, select the colored rectangle and press the Delete key.

At times, a user may want to see the kV level of screen objects directly from the oneline diagram, without having to open a text display. One way to do this is to color code the elements on the oneline diagram according to kV level. Right-click on the table and choose **Insert** from the local menu. Specify the Nom kV and color.

### Editing Default Field Positions ("Pos1", "Pos2", etc...)

Many objects allow you to specify some default fields to be automatically added when a new object is drawn. For example, you may want to insert the bus name next to all new Bus Objects. For objects that allow the insertion of fields, a tab with field position diagrams will be available when that object type is selected. In order to specify the default positions of fields, click on the tab showing the field positions and click on the table row corresponding to the Nom kV of the defaults of interest. For object types that do not allow specifying the defaults by Nom kV, simply update the given position diagram. The position diagram will show the present settings for the new fields. Positions that have a default specified will be highlighted and the name of the default field will be shown. Positions with no default field will not be highlighted and will say "Pos1", "Pos2", etc.... For objects that have more than one possible orientation (e.g. generators can be up, right, left or down), there are multiple position diagrams showing the positions for each orientation.

In order to modify the settings, move your mouse over the position on the dialog you want to change and click. This will bring up the appropriate Field Options Dialog such as the [Bus Field Options](#), [Generator Field Options](#), [Load Field Options](#), [Switched Shunt Field Options](#), [Line Field Options](#), [Interface Field Options](#), [Substation Field Options](#), [Area Field Options](#), [Super Area Field Options](#), or the [Zone Field Options](#). Simply select the field you want and choose **OK**. If you would like to set a default field to "none", click **Remove Field** instead of **OK**. You can also modify default fields by double-clicking on the table for "Pos1", "Pos2", etc... This brings up the appropriate Field Options Dialog as well. If you press the Delete key while on a "Pos" field in the table, it will set the default field to "none".

### Editing Stub Size and Stub Space

When utilizing the [Automatically Insert Transmission Lines](#) feature, the **Stub Size** and **Stub Space** values are used. Simulator will draw each automatically inserted branch such that it emerges from both its terminal buses at right angles. To accomplish this, Simulator draws each automatically inserted branch in three segments: two stubs perpendicular to the terminal buses having a length specified by the value supplied for **Stub Size**, and a third segment joining the two stubs. The amount of space between lines as they converge into a bus is set by **Stub Space**.

If you do not want transmission line stubs to be inserted, then set **Stub Size** to *none* or to a negative number. Note in order to set a value for **Stub Space**, **Stub Size** must be set to a positive number first.

### Editing CB Size

When you insert new transmission lines or transformers, circuit breakers will automatically be inserted on the branch with their size specified by CB Size. If you do not want Circuit Breakers inserted for a particular voltage level, then set CB Size to *none* or a negative number.

### Editing Pie Size

When you insert new transmission lines or transformers, pie charts will automatically be inserted on the branch with their size specified by Pie Size. If you do not want pie charts inserted for a particular voltage level, then set Pie Size to 'none' or a negative number.

### Set Default Font

Click on this button to set the default font used for new text fields. This is the font which will be used for all new text fields. Note however that the default font size set using this dialog is not used *unless* the option **Use the default font size for new text objects** is selected. Otherwise, the specific kind of object and voltage level specifies what the font size should be. For example, in the Interface tab, one of the defaults is Font Size. New Interface Fields will use this font size.

### Only Cut/Copy Display Objects, Not Power System Records

When you cut or delete an object from the online, Simulator needs to know whether you simply want to delete the display object from the online or to purge the definition of the power system object from the model. Check this box to tell Simulator to assume that it should always delete objects just from the online display, not from the power system model.

### Online / Bus View Background Color

To change the default online background color, click on the Change button to select a new color. This color applies to both new online diagrams and the background color of the [bus view online](#) diagram.

### Recommended Multi-KV Level Defaults

Click this button to change the default options back to Simulator recommended values. Multiple sets of default options will be specified based on voltage level for objects for which it is beneficial to do this (i.e. transmission lines).

### Recommended Single-KV Level Defaults

Click this button to change the default options back to Simulator recommended values. A single set of default options will be specified for each object type.

---

## Bus Selection Page

---

The Bus Selection page is currently used with two displays:

- The [Equivalencing Display](#), to partition the system into the study subsystem ("Study") and the external subsystem ("External").
- The [Facility Analysis Dialog](#), to determine the buses in the external subsystem from which the buses in the facility would be isolated.

The following description concerns the Equivalencing Display, but the functionality of the Bus Selection Page is similar for both applications.

### Buses, Areas, and Zones lists

For both equivalencing and facility analysis, use these lists to manually change the system designation of individual buses, areas, or zones. Note that changing the system designation on the area or zone tab is just another form of changing the designation of study / external system of individual buses. Click on the **Which System?** field in each of these tables to toggle the object's affiliation with the study or external systems, subject to the values of the **Filter by kV** option and the **Include how many tiers of neighbors?** field.

In addition, the bus selection table for [facility analysis](#) contains one additional column labeled **Selected**. This Yes or No field indicates which buses in the Study system belong to the Facility being analyzed. Double-clicking one of the fields in this column will toggle the value of the field between *Yes* and *No*.

### External

Use these fields to specify a [range of areas, zones, or buses](#) to be added to the external system. The **Filter by kV** and **Include how many tiers of neighbors?** controls will also shape the selection of buses to add to the external system.

### Study

Use these fields to specify a [range of areas, zones, or buses](#) to be added to the study system. The **Filter by kV** and **Include how many tiers of neighbors?** controls will also shape the selection of buses to add to the study system.

### Include how many tiers of neighbors?

This value indicates the number of tiers of neighbors to carry with each selected bus when adding the selected bus to either the study or external system. For example, if Neighbor Tiers is 1 and we elect to add bus X to the external system, both bus X and its first tier of neighbors will be added to the external system. If Neighbor Tiers is 0, only bus X will be added to the external system.

### Filter by kV

If the **Filter by kV** box is checked, then only buses having a nominal voltage level between the values given in the **Max kV** and **Min kV** fields can be selected.

### Set All As External

Click this button to assign all buses to the external system.

### Set Branch Terminals External or Study

Clicking either of these two buttons allows setting advanced filter criteria which define a branch or group of branches. Once the criteria are set, clicking **Filter** will select the terminal buses of all branches meeting the filter criteria, and set those buses to either the External or Study system, depending on which button was pressed.

### Select Buses using a Network Cut

A custom [network cut](#) can be defined for choosing which buses should remain in either the Study or External system.

### Save Buses to File

Once the system has been partitioned, this command allows you to store the numbers of the buses of the study system in a text file.

### Load Buses from File

Click on this button to load a listing of the buses to be included in the study system from a text file. You will be prompted to select the text file. The format of this text file is such that one bus number occupies each line. Any buses not identified in this file are defined as being in the external system.

### Example

Assume you would like to create an equivalent containing all the buses in areas 1-5 and 10, plus any tie buses, and bus number 2050.

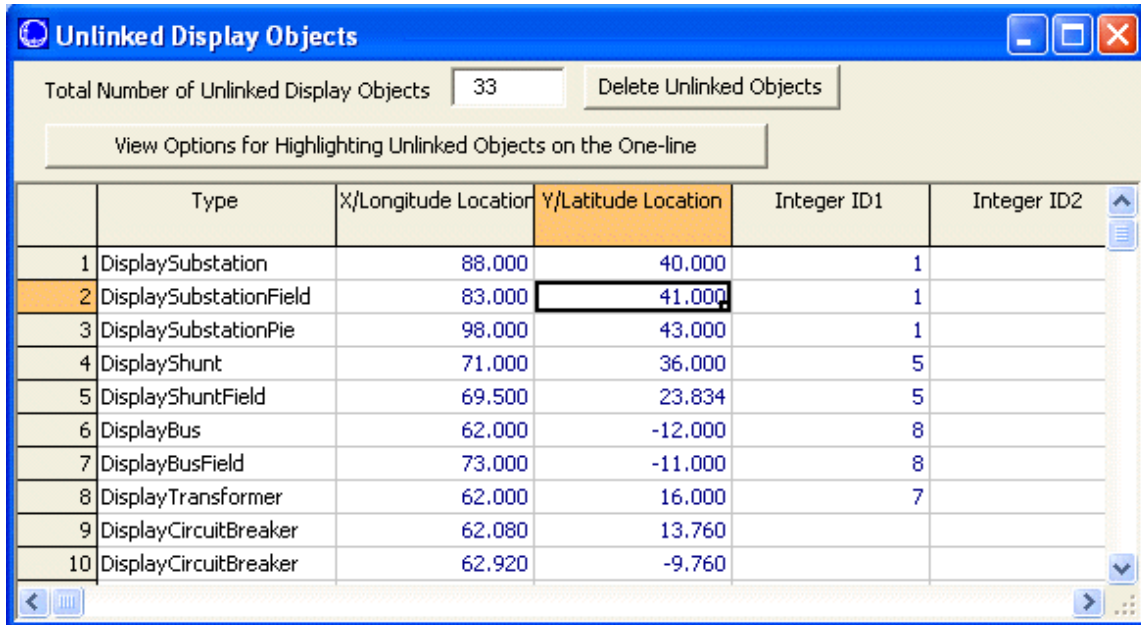
- Since initially all buses are in the study system, first enter 1-1000 in the **Areas** field of **Add to External System**. Since the area of every bus is within this range, this places all the buses in the external system. Alternatively, click **Set All as External** to accomplish the same objective.
- Set **Include how many tiers of neighbors?** to 1. This indicates that all subsequent selections will affect the specified buses and their first tier of neighbors.
- In the **Areas** field of **Add to Study System**, enter 1-5,10. This places all the buses in these areas, plus any tie buses (since Neighbor Tiers is 1), into the study system.
- In the **Buses** list, double-click in the **Which System** field for bus 2050 to change its status from *External* back to *Study*.



## List Unlinked Display Objects

An unlinked object is a display object not linked to a record in the power system model (model object). The existence of unlinked display objects on a oneline diagram can be misleading because they have zero flows and zero bus voltages associated with them. See [Object Relationships](#) for more information.

Go to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **List Display > Unlinked Display Objects** from the main menu in Edit Mode to identify unlinked objects.



**Unlinked Display Objects Dialog**

### Total Number of Unlinked Display Objects

Indicates the total number of unlinked objects on the display. Ideally, this number should be zero. If nonzero, the unlinked objects are identified in the table by type, screen location, and zoom range over which the object is visible.

### Delete Unlinked Objects

Click on this button to permanently remove all unlinked objects from the display. Exercise this option carefully. Generally, you will want to do this either when you have substantially modified a power flow case, such as by creating an equivalent, or when you are using the oneline with a new power system case.

### View Options for Highlighting Unlinked Objects on the Oneline

Click on this button will open the [Grid/Highlight Unlinked Objects](#) tab of the [Online Display Options Dialog](#).

### Type, X/Y Location

The remainder of the display shows the type, location, identification, layer, applicable zoom level, anchored property, and font size for each unlinked object. This table is a type of [Case Information Display](#) and thus behaves similarly to all other case information displays. Right-click on a record in this table to invoke the local menu. Select **Pan to Object on Open Onelines** to locate and select the unlinked object on the oneline diagram.



---

## Set Selected Field

---

The Set Selected Field option is an often unused featured in Simulator, but can at times prove very useful. Each object in Simulator has a property called **Selected**, which does nothing in Simulator other than allow the user to choose a specialized set of objects for some other purpose, such as [advanced filtering](#) a display. The user can add a column to most types of [case information displays](#) which shows the value of the **Selected** property for the type of object being observed. By default, this field is always set to *No*. However, the user can change the value of this field, and then sort the column, filter the display, or any other action that can normally be performed on a column of a Yes/No type.

In addition to modifying the **Selected** property manually in a [case information display](#), it is also possible to define a group of buses' **Selected** property by defining a [network cut](#) on the system. Choosing the **Set Selected Field for Network Cut** option from the **Tools** menu will open the [network cut](#) dialog automatically for setting the **Selected** property of a group of objects according to a desired cut plane chosen.

---

## Network Cut

---

The Network Cut tool is another method for choosing sets of buses for such features as [Scaling](#) or [Equivalencing](#).

The use of the network cut method is to define a set of branches as the "cut" plane, then choose a bus on one side or the other of the cut to indicate which side of the cut you are interested in. The network cut dialog provides the functionality necessary for defining the network cut.

### Defining Network Cut

The first step in defining the network cut is to choose the branches which define the cut plane in the system. The one caution is to be sure you select a closed loop cut plane. In other words, you must select a set of branches which completely topologically separates two portions of the system. If you only choose a partial cut plane, which does not completely cut the system in two distinct pieces, then attempted use of the ill-defined cut plane will fail.

To choose the branches which define the cut plane, simply highlight the branches by clicking on them in the list. For each branch you highlight, you must click on arrow button (pointing to the box on the right) to add the branch to your cut plane definition. You can make use of the control (Ctrl) and shift keys to select multiple branches at one time in the list of branches. Note that you can also select a cut plane similarly using interfaces or DC lines.

Once you have the branches (or interfaces or DC lines) selected which form the cut plane, you then need to choose a bus on either side of the cut plane to indicate which side you are interested in. The bottom panel on the display allows you to locate and select this bus.

### Require Paths to be Energized

This option can be checked if the branches forming a network cut should be energized to be included.

### Include How Many Tiers of Neighbors

Once the network cut has been defined by a set of branches and the bus defining which side of the cut is being examined has been chosen, this option will then include buses within so many tiers of the network cut boundary, on the opposite side of the cut as the specified bus. If the number of tiers is set to zero, then the buses examined will only be those on the same side of the cut as the bus selected.

### Setting the Field

Once the network cut has been defined and the side of the cut to be examined has been chosen, the appropriate field for each bus in the area of interest can be set. If the network cut is for the equivalencing tool, then the field to set is the **Which System** property, which defines if a bus is in the external or study system when creating an equivalent. If the network cut is for the scaling tool, then the field to set is the **Scale** property, which defines if a bus is to be included in a scaling action.

### Filter by KV

The buses within the network cut can be filtered by nominal voltage level before the field(s) are set. Only buses that are within the network cut and are within the nominal KV level specified will have their fields set to the desired value.

### Use Area/Zone Filters

The buses within the network cut can be filtered by the area or zone in which they are contained before the field(s) are set. Only buses that are within the network cut and

whose area meets the defined Area/Zone filter specified will have their fields set to the desired value.

---

## Browse Open Onelines

---

All the objects in all the open .pwd files can be listed in the Browse Online Environment dialog. To do so, go to the [Onelines](#) ribbon tab, [Active](#) ribbon group and choose **List Display > Browse open onelines** in Edit Mode or Run Mode. The power system objects are listed on tabs according to their type. For instance, all the buses and bus fields are listed in the **Buses** tab while all the generators and generator fields are listed in the **Generators** tab. The **Others** tab lists all the objects on the onelines which are not associated with power system elements, e.g. background lines or images. The **All** tab lists all the objects in all the open onelines. The default display columns for each grid include the oneline file name of the object and the type of the object. Objects that are linked to case elements also have identifying information about the data element that they represent. The **Others** and **All** pages include the default display columns for generic display objects.

The oneline displays can be panned to the selected object. To do so, select one object from the Browse Online Environment dialog and right click to invoke the local popup menu. Click **ScreenObject Records > Pan to Object on Open Onelines**. This will bring the oneline with the selected object to the front and pan to the selected object. When panning, the browsing dialog can be kept on top if **Keep browsing dialog on the top when choosing Pan Oneline to Object** is checked. Otherwise, the dialog will be sent to back when panning.

---

## Unused Bus Numbers

---

When making changes to a power system case, it is often beneficial to know which bus numbers are still available. To save a list of unused bus numbers to a text file, while in [Edit Mode](#), go to the [Tools](#) ribbon tab, and choose **Modify Case > List of Unused Bus Numbers** from the [Edit Mode](#) ribbon group. Enter a [range of bus numbers](#) in the dialog and click **OK**. In the Save As Dialog, specify the text file where the list of bus numbers should be written and click **Save**. Only those unused bus numbers in the specified range will be listed in the file. Clicking **Cancel** at any point will abort saving the list to a file.

---

## Palette Overview

---

The display object palettes are designed to help you lay out a new oneline diagram for a pre-existing power flow model as quickly as possible. Display object palettes exist for [Areas](#), [Buses](#), [Substations](#) and [Zones](#). Palettes list the display objects that you have already added to the oneline, the display objects that have not yet been drawn that neighbor those displayed objects, and the set of all display objects that have not yet been added to the drawing. By selecting and dragging a display object name from either the **Undisplayed Neighbors** list or the **All Undisplayed** list to a location on the oneline diagram where you would like that object to appear, you can add that display object to the drawing with very little effort.

To see which elements neighboring a particular device are already on the oneline diagram and which are not, highlight a device in either the **Displayed** or **All Undisplayed** list. The **Displayed Neighbors** and **Undisplayed Neighbors** columns will list the corresponding neighboring devices for the selected device, allowing you to drag the undisplayed neighbors to the diagram if you wish.

The display object palettes, in conjunction with the of other devices, are especially useful for adding a large region of an interconnection in relatively little time.

To display one of the four available [palettes](#), select **Show Insert Palette For** from the [Quick Insert](#) ribbon group on the [Draw](#) ribbon tab. To display the palette with the focus set to an object already displayed, right-click on that object on the diagram and select **\*\*\* Palette** (where \*\*\* is the name of the object type) from the popup menu.



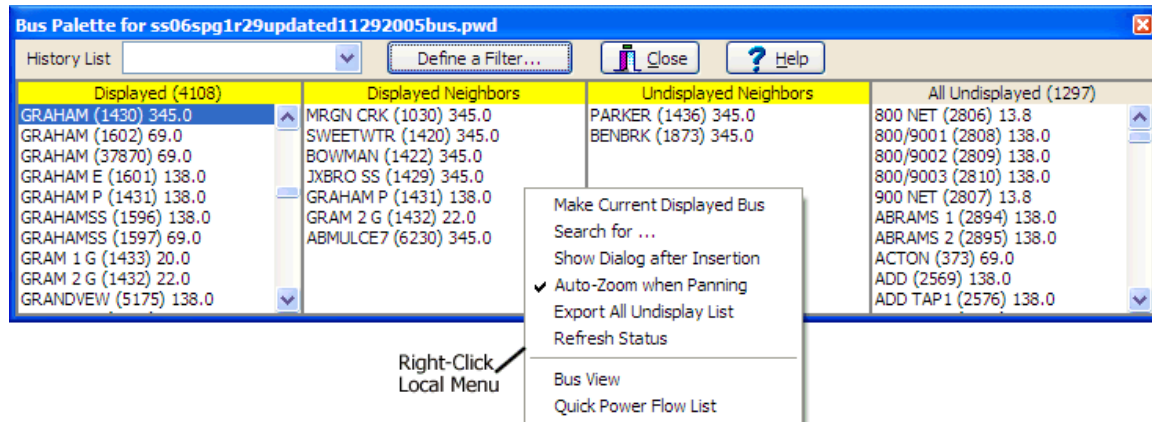
## Using the Insert Palettes



The display object palettes are designed to help you lay out a new oneline diagram for a pre-existing power flow model as quickly as possible. Display object palettes exist for [Areas](#), [Buses](#), [Substations](#) and [Zones](#). To open one of the palettes you must be in [Edit Mode](#) and then go to the [Draw](#) ribbon tab, and choose the **Show Insert Palette For Menu** from the [Quick Insert](#) ribbon group.

The Insert Palettes can be used in conjunction with the various auto-insert capabilities also found on the [Quick Insert](#) ribbon group. These features allow for the creation of oneline diagrams much more quickly than inserting oneline objects one at a time using the [Individual Insert](#) ribbon group.

When you choose one of the palettes the dialog which appears will look like the image below. The following image shows the Bus Palette.



Palettes list the display objects that you have already added to the oneline, the display objects that have not yet been drawn that neighbor those displayed objects, and the set of all display objects that have not yet been added to the drawing.

The display object palettes feature the following controls:

### Displayed

Lists those objects defined in the power flow case that have already been added to the oneline diagram. When you click on an entry in the **Displayed** list, the contents of the **Displayed Neighbors** and **Undisplayed Neighbors** lists will be refreshed to identify all objects that neighbor the display object you selected. Only the display objects matching the type of Insert Palette opened will be shown in the list.

Double click on an entry in this list to pan to it so that the selected display object appears in the center of the screen. If you are looking at the Insert Palette for Buses, you can right-click on an entry and choose either [Quick Power Flow List](#) or [Bus View](#) to learn more about the selected bus.

### Displayed Neighbors

Lists those objects defined in the power flow case that neighbor the display object selected in the **Displayed** list that have already been drawn on the oneline diagram. You can make a "Displayed Neighbor" the selected "Displayed" object by right-clicking on a display object in

the **Displayed Neighbor List** and selecting **Make Current Displayed** from the local menu. Only the display objects matching the type of Insert Palette opened will be shown in the list.

### Undisplayed Neighbors

Lists those display objects defined in the power flow case that neighbor the object selected in the **Displayed** list that have not yet been drawn. To add the undrawn display object to the oneline diagram, select its name with the left mouse button and keep the left mouse button pressed as you move the mouse to the point on the oneline where you would like to drop the object. When you let go of the left mouse button, the **Information Dialog** box for that display object will appear. (If you do not want the information Dialog box to appear the use the right-click local menu to uncheck the option to regarding showing the information dialog). Use the Information Dialog to change display parameters for the object and click **OK** to finish dropping the object onto the oneline. The display object you have just added will be appended to the end of the **Displayed** list and will also be added to the **History List** so that you can identify its own undrawn neighbors quickly. Only the display objects matching the type of Insert Palette opened will be shown in the list.

If you are looking at the Insert Palette for Buses, you can right-click on an entry and choose either [Quick Power Flow List](#) or [Bus View Display](#) to learn more about the selected bus.

### All Undisplayed

Lists those display objects defined in the power flow case that have not yet been added to the oneline diagram, regardless of whether the object neighbors a displayed object or not. This list functions identically to the **Undisplayed Neighbors List**. To add the undrawn display object to the oneline diagram, select its name with the left mouse button and keep the left mouse button pressed as you move the mouse to the point on the oneline where you would like to drop the object. When you let go of the left mouse button, the **Information Dialog** box for that display object will appear. (If you do not want the information Dialog box to appear the use the right-click local menu to uncheck the option to regarding showing the information dialog). Use the Information Dialog to change display parameters for the object and click **OK** to finish dropping the object onto the oneline. The display object you have just added will be appended to the end of the **Displayed** list and will also be added to the **History List** so that you can identify its undrawn neighbors quickly. Only the display objects matching the type of Insert Palette opened will be shown in the list.

If you are looking at the Insert Palette for Buses, you can right-click on an entry and choose either [Quick Power Flow List](#) or [Bus View Display](#) to learn more about the selected bus.

### History List

Identifies the display objects you have added to the oneline diagram using the insert palettes since the last time you opened the display object palette. Select a name from this list to display the undrawn neighbors of the corresponding display object. Only the display objects matching the type of Insert Palette opened will be shown in the list.

### Define a Filter

The **Define a Filter** button opens the [Advanced Filters](#) dialog. This dialog allows you to customize which display objects appear in the various lists. This can be helpful, for example, if you wish to add objects to the oneline that reside only in particular areas, or if you don't want objects less than a certain voltage level to be listed.

When you click **OK** on the [Advanced Filters](#) dialog, the display object palette's lists will automatically be updated to reflect the filter settings.

### Close

Click **Close** when you are done using the display object palette.

**Right-Click Local Menu of the Palette**

There are several options available on the Right-Click

**Search for** : Click this option to search through the lists for a particular object. It will open the familiar [Find Object dialog](#).

**Auto-Zoom When Panning** : When double-clicking on the displayed objects in the palette, you will automatically be panned so that object is in the center of the screen. Check this option to also automatically zoom in/out to a reasonable level at the same time.

**Export All Undisplayed List** : Choose this option to export this list of undisplayed objects to a text file.

**Refresh Status** : Choose this option to ensure that the list of Displayed/Undisplayed objects is correct.

**Show Dialog after Inserting** : check or uncheck this option to control whether the information dialog for the object will appear when using the palette

---

## Automatically Inserting Buses

---

Simulator can automatically insert buses on your oneline diagram if you have data regarding their spatial or geographic location. To achieve this, go to the [Draw](#) ribbon tab and choose **Auto Insert > Buses** on the [Quick Insert](#) ribbon group. This opens the Auto Insert Bus Dialog.

**Note:** Auto-insert buses is only for inserting bus objects on the diagram representing existing data. In other words, you cannot use the auto-insert buses routine to add new buses to the load flow model. See topics on loading data from [auxiliary files](#) or from [Excel](#) for creating new buses (or other objects) in an existing load flow case.

### Insert by longitude, latitude

If you have stored latitude and longitude information with the Bus records in Simulator, you can use that information to automatically insert the buses on a oneline diagram. Select the option on the dialog for finding bus locations based on **Longitude and Latitude stored with data records**. When selected, the Map Projection option will become enabled. Choose the type of projection you would like to be used for placing the buses on the diagram.

### Insert by locations specified in a file

You must first specify the file which contains the location data. You must also specify whether the file contains x,y coordinates or longitude, latitude coordinates. The format of the location data text file is as follows:

The first line of this text file is ignored by Simulator. The following lines consist of three numbers: Bus Number, X location, and Y location. If you are reading longitude, latitude, then X signifies longitude and Y signifies latitude.

Num, X Location, Y Location

1, 24001.46, 19715.15

3, 24001.46, 19715.15

16, 24130.91, 19638.99

17, 24007.31, 19093.09

21, 23649.27, 18439.07

22, 23649.27, 18439.07

etc...

Simulator will place the buses on the oneline diagram at the X, Y locations given. If you specified that the file contained longitude, latitude information, then select the **Map Projection** to use when converting the longitude, latitude values from the file. The buses will be drawn according to default bus object information defined in the Default Drawing Values for New Objects dialog.

If you would like Simulator to **Autoinsert transmission lineswhen finished** with the auto insertion of your buses, check the box. Transmission lines and transformers will be automatically drawn based on the Line/XFMR default options defined in the [Default Drawing Values for New Objects](#) dialog.

Check the **Insert bus only if not already shown on the oneline** to ensure that multiple display objects are not inserted for the same element.

Clicking the **OK** button will instruct Simulator to continue by placing the buses according to the specified settings on the dialog. Click **Cancel** to exit the process without inserting the buses.

---

## Automatically Inserting Transmission Lines

---

The Automatic Insertion of Lines/Transformers Dialog is used to automatically draw transmission lines and transformers on the oneline diagram between existing bus display objects. Only branches that are already defined in the power flow case can be added automatically; if you need to define a brand new branch, see [Transmission Line Display Objects](#). Thus, this option is useful only when you are starting with an existing power flow case, not building a case from scratch.

To insert lines and transformers automatically, you must first have drawn the buses for each end of the device. Simulator then draws the branch display objects connecting the buses for each transmission line/transformer in the power flow case not already shown on the oneline.

To display the dialog, in the Edit Mode go to the [Draw](#) ribbon tab and choose **Auto Insert > Lines** on the [Quick Insert](#) ribbon group.

The dialog has the following options:

### Minimum kV Level

Simulator will automatically draw line and transformer display objects between terminals whose nominal voltages meet or exceed the minimum kV level specified in this field. For a branch object to be drawn, either one of its terminals must satisfy this criterion. This option is useful for suppressing the automatic addition of generator step-up transformers if that kind of detail is not warranted.

### Default Drawing Values

This button opens the [Default Drawing Values Dialog](#). Options such as the automatic insertion of transmission line stubs, text fields, circuit breakers, and pie charts are specified in the [Default Drawing Values Dialog](#).

### Insert Text Fields

If this box is checked, the default fields associated with the transmission line will appear around the transmission line.

### Insert Equivalenced Objects

If this option is checked, [equivalenced](#) objects modeled as lines will also be automatically inserted with the real transmission lines.

### Use Only Selected Buses

Instead of having Simulator automatically insert line and transformer display objects throughout the oneline diagram, you can force it to insert the new objects only between the [bus display objects](#) that are currently selected. This option is enabled only when two or more bus display objects are currently selected on the oneline.

### Insert Pie Chart for Lines with No Limit and Bus Ties

If this option is checked, Simulator will add [pie chart objects](#) to the lines that have no given limit or are bus ties as they are auto-inserted. Typically if a line is a bus tie or has no given limit, it is meaningless to include a pie chart on the element, since no relevant information about the transmission element can be gained from the pie chart object.

### Insert Multi Section Lines

Check this option if you wish for Simulator to automatically insert lines designated as [multi-section lines](#) as well.

### Identifying Bus Ties

This area deals with lines used as ties between breakers. These lines are modeled as zero impedance connections. The identification of a branch as a bus tie depends on the value specified as the **Maximum P.U. Impedance for Bus Ties**. Branches with total P.U. impedance below this value will be considered bus ties when auto-inserted.

In this area two choices are given for how to insert the bus tie breakers: **Do not insert stubs for bus ties** and **Only insert a single circuit breaker**. The first choice allows you to decide if line stubs will be drawn when the tie breaker is inserted. The second sets whether or not only one circuit breaker is inserted on the tie breaker. This could be useful for determining real lines from bus tie breakers.

---

## Automatically Inserting Generators

---

The Automatic Insertion of Generators Dialog is used to automatically draw generators on the oneline diagram on existing bus-display objects. Only generators that are already defined in the power flow case can be added automatically; if you need to define a new generator, see [Generator Display Objects](#). Thus, this option is useful only when you are starting with an existing power flow case, not building a case from scratch.

To insert generators automatically, you must first have drawn the terminal bus for each device. Simulator then draws the generator display objects connected to the buses for each generator in the power flow case not already shown on the oneline.

To display the dialog, in the Edit Mode go to the [Draw](#) ribbon tab and choose **Auto Insert > Generators** on the [Quick Insert](#) ribbon group.

The dialog has the following options:

### Minimum kV Level

Simulator will automatically draw generator display objects at terminal buses whose nominal voltages meet or exceed the minimum kV level specified in this field. For a generator object to be drawn, its terminal must satisfy this criterion.

### Default Drawing Values

This button opens the [Default Drawing Values Dialog](#). Options such as the automatic insertion of text fields are specified in the [Default Drawing Values Dialog](#).

### Insert Text Fields

When this box is checked, the default fields associated with the generator will appear around the generator, such as voltage, name, and/or MW.

### Insert Equivalenced Objects

This field is not used when automatically inserting generators.

### Use Only Selected Buses

Instead of having Simulator automatically insert generator display objects throughout the oneline diagram, you can force it to insert the new objects only between the bus display objects that are currently selected. This option is enabled only when two or more bus display objects are currently selected on the oneline.



---

## Automatically Inserting Loads

---

The Automatic Insertion of Loads Dialog is used to automatically draw loads on the oneline diagram on existing bus display objects. Only loads that are already defined in the power flow case can be added automatically; if you need to define a brand new load, see [Load Display Objects](#). Thus, this option is useful only when you are starting with an existing power flow case, not building a case from scratch.

To insert loads automatically, you must first have drawn the terminal bus for each device. Simulator then draws the load display objects connected to the buses for each load in the power flow case not already shown on the oneline.

To display the dialog, in the Edit Mode go to the [Draw](#) ribbon tab and choose **Auto Insert > Loads** on the [Quick Insert](#) ribbon group.

The dialog has the following options:

### Minimum kV Level

Simulator will automatically draw load display objects at terminal buses whose nominal voltages meet or exceed the minimum kV level specified in this field. For a load object to be drawn, its terminal must satisfy this criterion.

### Default Drawing Values

This button opens the [Default Drawing Values Dialog](#). Options such as the automatic insertion of text fields are specified in the [Default Drawing Values Dialog](#).

### Insert Text Fields

When this box is checked, the default fields associated with the load will appear with the loads, such as MVAR and/or MW.

### Insert Equivalenced Objects

If this option is checked, [equivalenced](#) objects modeled as loads will also be automatically inserted with the real transmission lines.

### Use Only Selected Buses

Instead of having Simulator automatically insert load display objects throughout the oneline diagram, you can force it to insert the new objects only between the bus display objects that are currently selected. This option is enabled only when two or more bus display objects are currently selected on the oneline.

---

## Automatically Inserting Switched Shunts

---

The Automatic Insertion of Switched Shunts Dialog is used to automatically draw switched shunts on the oneline diagram on existing bus-display objects. Only switched shunts that are already defined in the power flow case can be added automatically; if you need to define a brand new switched shunt, see [Switched Shunt Display Objects](#). Thus, this option is useful only when you are starting with an existing power flow case, not building a case from scratch.

To insert switched shunts automatically, you must first have drawn the terminal bus for each device. Simulator then draws the switched shunt display objects connected to the buses for each switched shunt in the power flow case not already shown on the oneline.

To display the dialog, in the Edit Mode go to the [Draw](#) ribbon tab and choose **Auto Insert > Switched Shunts** on the [Quick Insert](#) ribbon group.

The dialog has the following options:

### Minimum kV Level

Simulator will automatically draw switched shunt display objects at terminal buses whose nominal voltages meet or exceed the minimum kV level specified in this field. For a switched shunt object to be drawn, its terminal must satisfy this criterion.

### Default Drawing Values

This button opens the [Default Drawing Values Dialog](#). Options such as the automatic insertion of text fields are specified in the [Default Drawing Values Dialog](#).

### Insert Text Fields

When this box is checked, the default fields associated with the switched shunt will appear around the switched shunt, such as nominal MVAR.

### Use Only Selected Buses

Instead of having Simulator automatically insert switched shunt display objects throughout the oneline diagram, you can force it to insert the new objects only at the bus display objects that are currently selected. This option is enabled only when two or more bus display objects are currently selected on the oneline.

---

## Automatically Inserting Interface Display Objects

---

The Automatic Insertion of Interfaces Dialog is used to automatically insert [interface objects](#) on the oneline diagram between existing [area/zone display objects](#). [Interface pie chart objects](#) can also be inserted as part of this process. The automatic insertion of interface display objects greatly accelerates the construction of interface diagrams, which are particularly useful for animating the results of [PTDF](#) calculations. Area-area and zone-zone interface records can be displayed using interface display objects as well as interfaces comprised only of transmission branches.

Note that inserting interface objects on a oneline diagram does NOT add interface objects to the load flow case. The interface definitions need to be added to the load flow case prior to automatically inserting the graphical interface objects on a oneline diagram. The quickest way to add area to area or zone to zone interface definitions in the load flow case is to [automatically insert the definitions](#) in the interface case information display.

Once you have the interface definitions defined in the case, you can automatically insert the interface objects on a diagram using the following procedure:

- On a oneline diagram (either an already open diagram or a brand new one created by choosing **New Oneline** from the [application button](#)), place [area/ zone objects](#) at the desired locations if inserting area-area or zone-zone interface records. Otherwise, the terminal buses for the branch elements in the interface should be inserted at the desired locations on the oneline diagram.
- Go to the [Draw](#) ribbon tab and choose **Auto Insert > Interfaces** on the [Quick Insert](#) ribbon group.
- Check the **Insert Pie Charts on Interfaces** box to automatically insert interface pie chart objects when the interfaces are inserted. If this option is selected, change the **Default Size of Interface Pie Charts** to specify their size.
- Choose the type of interfaces to insert. **Area to Area or Zone to Zone Interfaces** and **Line/Transformer Interfaces** can be inserted. Area-area and zone-zone interfaces will be drawn between the respective area and zone objects displayed on the oneline. Line/transformer interfaces will be drawn based on the average location of the terminal buses of all transmission lines comprising the interface. If inserting line/transformer interfaces, the option **Minimum Length of Line/Transformer Interfaces** can be set to prevent interfaces that are too short from being inserted.
- Select **OK** to insert the new oneline objects. New interface objects are automatically inserted based on the selected criteria for any corresponding interface record that is not already represented.

Note that you can do this automatic insertion as often as you like. The **Number of Interfaces Not Shown** field indicates how many interfaces still need to be added to the diagram to represent all defined area-area, zone-zone, or line/transformer interfaces. It is a read-only field.

---

## Automatically Inserting Substations

---

Simulator can automatically insert substations on your oneline diagram if you have data regarding their spatial or geographic location. To achieve this, go to the [Draw](#) ribbon tab and choose **Auto Insert > Substations** on the [Quick Insert](#) ribbon group. This opens the Auto Insert Substation Dialog.

### Insert by longitude, latitude

If you have stored latitude and longitude information with the Substation records in Simulator, you can use that information to automatically insert the substations on a oneline diagram. Select the option on the dialog for finding substation locations based on **Longitude and Latitude stored with data records**. When selected, the [Map Projection](#) option will become enabled. Choose what type of projection you would like to be used for placing the substations on the diagram.

### Insert by locations specified in a file

You must first specify the file which contains the location data. You must also specify whether the file contains x,y coordinates or longitude, latitude coordinates. The format of the location data text file is as follows

The first line of this text file is ignored by Simulator. The following lines consist of three numbers: Substation Number, X location, and Y location. If you are reading longitude, latitude then X signifies longitude and Y signifies latitude.

Num, X Location, Y Location

1, 24001.46, 19715.15

3, 24001.46, 19715.15

16, 24130.91, 19638.99

17, 24007.31, 19093.09

21, 23649.27, 18439.07

22, 23649.27, 18439.07

etc...

Simulator will place the substations on the oneline diagram at the X, Y locations. If you specified that the file contained longitude, latitude information, then select the [Map Projection](#) you would like to be used when converting the longitude, latitude values from the file. The substations will be drawn according to default substations object information defined in the Default Drawing Values for New Objects dialog.

Finally, if you would like Simulator to **Auto insert transmission lines when finished** with the auto insertion of your substations, check this box. Transmission lines and transformers will be automatically drawn based on the Line/XFMR default options defined in the [Default Drawing Values for New Objects](#) dialog.

Check the **Insert substation only if not already shown on the oneline** to ensure that multiple display objects are not inserted for the same element.

Clicking the **OK** button will instruct Simulator to continue by placing the substations according to the specified settings on the dialog. Click **Cancel** to exit the process without inserting the substations.

---

## Automatically Inserting Borders

---

PowerWorld Simulator allows you to automatically insert geographic borders, which PowerWorld Corporation has drawn. These include the states in the United States of America and several international borders as well. You may also define a border in a text file and insert this User-Defined border. It is highly recommended that you create a diagram and insert borders before you begin adding power system objects to the diagram.

To bring up the **Auto Insert Borders Dialog**, go to the [Draw](#) ribbon tab and choose **Auto Insert > Borders** on the [Quick Insert](#) ribbon group.

### Options

The Options tab allows you to set characteristics of the border lines when they are inserted on the oneline diagram. Most of these options are available from the Format menu in Edit Mode, and can be changed for a border by selecting the border(s) in Edit Mode and then selecting the appropriate option from the Format menu to change the desired option(s).

#### Make border lines immobile

This option is very useful for preventing you from inadvertently moving the border lines in Edit Mode once they have been placed.

#### Automatically link county and state borders to supplemental data

Background lines created from inserting borders can be linked to [Supplemental Data](#) records for purposes of further identifying these objects. This is useful when applying filters, applying dynamic formatting, using Select By Criteria, or defining custom hints with display objects created from inserting the borders. This option is only available when inserting United States borders.

When this option is checked, background lines created for the borders will be automatically linked to Supplemental Data records. If necessary, Supplemental Data records will be created with the Classification being either *State* or *County*, depending on the Border Type selected when inserting United States borders, and the Name being the name of either the state or county.

#### Line Options

Choose the line thickness and line color for the border lines.

#### Fill Options

Choose the fill options if you wish to "fill in" the border regions with a background fill color.

#### Stack Level

Choose the stack level for the border lines. The stack level affects which objects appear "over" or "under" other objects on the diagram. Typically background lines are either placed in the Background or Base levels, so that they appear beneath power system objects on the diagram. Further we would recommend that if you are using background fill colors for the borders that you choose a stack level of Base. The only difference between the Background and Base levels is that the Base level objects can be right-clicked on in Run Mode and allows the default diagram popup menu to appear. Right-clicking on Background level objects in Run Mode will not display the default popup menu.

#### Layers

When Borders are automatically inserted on a diagram using this tool, a *Borders* [screen layer](#) will be added to the layer drop-down box, if a *Borders* layer does not already exist for the oneline diagram. You can choose to leave the borders in the standard *Default* layer, to place the borders in the new *Borders* layer, or you can [create a new custom layer](#) by clicking the **Define Layers** button. If no default borders layer has been previously established (borders were previously inserted on the oneline), the *Borders* layer or selected user created layer will be set as the default Borders layer. The default Borders layer is different than the standard *Default* layer for objects. The default Borders layer information is stored in the system registry, and is only used to recall which layer borders were assigned to the previous time borders were automatically inserted.

### **Apply Default Drawing Values**

Click this button to apply the Background line settings from the [Default Drawing Values](#).

### **Border File Path**

Designate the border file location.

## **PowerWorld Library**

### **United States**

This tab allows you to select states in the United States that you want to insert. (Note: to select several states to insert at once, use the Ctrl and Shift keys while clicking with your mouse.) These states will be placed on the screen such that as you add new states they will be placed geographically appropriately.

Once you have selected the states you want to insert, Click OK.

### **Canada**

This tab allows you to select provinces in Canada that you want to insert. (Note: to select several provinces to insert at once, use the Ctrl and Shift keys while clicking with your mouse.) These provinces will be placed on the screen such that as you add new provinces they will be placed geographically appropriately.

Once you have selected the provinces you want to insert, Click OK.

### **World**

This tab allows to insert borders which PowerWorld Corporation has acquired for countries around the world. PowerWorld Corporation will continue to add more border files as we receive them. Note: if you have drawn a oneline which contains a geographic border you would like us to include in future version of Simulator, please contact us at [support@powerworld.com](mailto:support@powerworld.com) and we'll add it to our next release.

### **[Map Projection](#)**

This setting is important for anyone including border files outside of North America. By default, Simulator draws borders using a simple conic projection referenced to North America. This can result in border files of other countries being drawn incorrectly if based on this reference. To correctly draw borders of other countries around the world, select the Entire World setting, which uses a Mercator projection.

## **User-Defined Borders**

This tab allows you to read in a border from a text file that you create. The first row of this text file is a comment row and is ignored when reading it in. After this it reads in the description of each background line. This description starts with the number of points in the

line followed by a list of x, y coordinates. The file ends when the number of points for a line is read as -1. An example file follows:

Comment row

5

59, 60

10, 20

40.3, 95.20

89.3, 22.11

79.5, 34.56

45

40, 66

etc...

-1 this signifies the end of the file

To read the data, first specify the file name and location containing the data using the Browse button. Next indicate if the file contains data in Longitude, Latitude or in converted Simulator X,Y coordinates. If the data is Longitude and Latitude, then also specify if the map projection to use should be North America (simple conic) or Entire World (Mercator).

---

## Equivalents

---

An equivalent power system is a power system model of smaller dimension than the original system that approximates the behavior of the original system reasonably well. In reality, most power system models are actually an "equivalent" of a much larger interconnected network. When performing power system studies, it may be desirable to reduce the size of the system model even further so that it may be solved more quickly.

To bring up the [Equivalents Display](#), while in [Edit Mode](#), go to the [Tools](#) ribbon tab, and choose **Create Equivalent** from the [Edit Mode](#) ribbon group.

The most important part of constructing an equivalent is determining which buses should be explicitly retained in the equivalent, and which buses should be *equivalenced*, or removed from the case. Several definitions are useful here:

### Study System

The buses that are to be retained.

### External System

The buses that are to be equivalenced.

### Boundary Buses

Any buses in the study system that are connected to buses in the external system.

How well the equivalent system approximates the behavior of the original system depends upon which buses are retained in the study system. Retaining more buses yields results that more closely match those of the original case, but at the expense of greater computation time. The number of buses to retain in the study system depends upon how the equivalenced system will be used. Building system equivalents is as much an art as it is a science, with few solid rules of thumb. However, to improve accuracy, you should retain as many generator units as possible.

The actual equivalent is constructed by performing a matrix reduction on the bus admittance matrix. A result of this process is the creation of "equivalent" transmission lines that join boundary buses equipped with equivalent shunts or loads. Equivalent lines typically have a circuit identifier of *99*, but have also been seen to have other numerical values between 90 and 99, or an alphanumeric identifier of *EQ*. Since many of the equivalent lines created during the matrix reduction have very high impedance values, an option is provided to ignore equivalent lines with impedances exceeding a specified threshold value. Additionally, an option is provided to convert the equivalent shunts added at the boundary buses to constant PQ loads. These PQ loads will be given circuit identifiers similar to those given to equivalent transmission lines.



---

## Equivalents Display

---

The **Equivalents Display** is used to construct equivalent systems. An *Equivalent System* is a system of smaller dimension that exhibits similar power flow properties. Equivalent systems are constructed to help accelerate computation time without sacrificing a significant amount of accuracy. For more information, please see [Equivalents](#).

To bring up the Equivalents Display, while in [Edit Mode](#), go to the [Tools](#) ribbon tab, and choose **Create Equivalent** from the [Edit Mode](#) ribbon group. This display contains two pages, the **Bus Selection Page** and the **Create Equivalent Page**. Use the Bus Selection Page to partition the power system into the study system and the external system. Use the Create Equivalent Page to

- Save the external system in a file
- Extract the external system
- Build an Equivalent

Each of these tasks is described below.

### Bus Selection Page

To perform any of the tasks described on the Create Equivalent Page, you first need to specify the study system and the external system. Do this by directly assigning buses to the desired system. The [Bus Selection Page](#) has been designed to provide a number of powerful and complimentary ways of accomplishing this task.

The most important point to keep in mind when using this page is that membership in the study system and the external system is on a bus-by-bus basis (as opposed to by areas or zones). Thus, each bus is either in the study system or the equivalent system. Each bus' current assignment is indicated in the Buses list, which is shown on the bottom left corner of the page. The Buses list is a [Case Information display](#) and can be manipulated similar to other displays. By default, all buses initially reside in the study system. Please see [Bus Selection Page](#) for more details.

### Create Equivalent Page

The Create Equivalent Page allows you to save the external system in a file, to delete the external system, and to build the power system equivalent.

#### Build Equivalent

This procedure constructs an equivalent system. For background on equivalents, please see [Equivalents](#). The following sections contain the options for building an equivalent.

#### Delete All External Generators or Retain Generators with Max MW Ratings Above

If the **Delete All External Generators** checkbox is checked, the equivalencing routine will remove all external generators from the case, regardless of their MW rating. Otherwise, the equivalencing routine will add to the study system any generators originally in the external system whose real power output exceeds the specified value. Retaining large generators often makes an equivalent significantly more accurate. If you do not wish to retain any additional generators, check the **Delete All External Generators** checkbox.

#### Retain Remotely Regulated Buses

Some generators and transformers regulate buses other than their terminals. When this box is checked, these remotely regulated buses are automatically included in the equivalent if the regulating generator or transformer is included. If the box is not checked, the regulated bus is set to the terminal of the retained object. It is strongly recommended that you leave this box checked at all times.

### **Retain Branch Terminals For**

This section allows you to customize the retention of branch terminals for special-case types of branches. You may choose to retain terminal buses for transformers, zero impedance ties, area tie lines, and/or zone tie lines.

### **Max Per Unit Impedance for Equivalent Lines**

During the equivalencing process, a number of equivalent lines are created joining the boundary buses. All equivalent lines with per unit impedance values above this threshold are ignored.

### **Two Character Circuit ID for New Equivalent Lines**

Choose the circuit identifier to be used for the equivalent lines that are created. Typical circuit IDs for equivalent lines are *99* or *EQ*.

### **Remove External Objects from Onelines**

This feature removes display objects associated with the external system from any open onelines.

### **Convert Equivalent Shunts to PQ Loads**

During the equivalencing process, shunt elements are added at the boundary buses. Check this box if you would like these equivalent shunts converted to constant PQ loads. If this option is checked, equivalent loads are created with a [load ID](#) of *99*.

### **Remove Radial Systems**

Checking this option results in all radial connections in the network to be reduced to their nearest non-radial bus (i.e. node.) The equivalencing routine will iteratively reduce the network when this option is checked, until no more radial connections exist in the system.

### **Delete Empty Areas/Zones/Substations that occur from Equivalencing**

Since equivalencing is a process which ultimately removes buses from the system, and Areas, Zones and Substations are system devices which are groups of buses, this option will automatically remove the definition of these types of objects when all buses within them are removed from the case during the creation of the equivalent.

Select **Build Equivalent System** to construct the equivalent system. Constructing an equivalent system permanently removes the external system from the case and adds a number of equivalent lines and shunts/loads.

### **Saving the External System in a File**

This procedure allows an external system to be saved in a file **without deleting** the external system. This option is useful for allowing you to save a portion of the system in a file, modify it using perhaps another program, and then use [Append Case](#) to append the modified file to the original case.

When saving the external case, there are two options: 1) save just the external case, or 2) save the external case and any ties to the original case. Option one just saves the external case, while option two saves the external case and any transmission lines or transformers that connect the external system to the rest of the system. Save just the external case if you are planning to use the external case as a standalone case. Save the external case and its ties if you are planning to modify the external case and then to append it back to the original case.

Click **Save External System** to save the external system in a file. You will be prompted for the desired [case format](#).

### **Deleting the External System**

This procedure deletes the external system from the original case. All devices in the external system are removed, including any buses and lines/transformers in the external system and any lines/transformers that join the external system to the study system. Check the **Remove External Objects from Onelines** to remove any display objects linked to the external system from the open onelines.

This command **permanently removes** the external system from the case; an equivalent system is **not** created. Select **Delete External System** to actually delete the system.

---

## Scaling

---

Use the **Power System Scaling Dialog** to scale the load, generation, or bus shunts uniformly for either the entire case or a group of selected buses. This display allows you to scale any of the following values:

- Bus real power load
- Bus reactive power load
- Generator real power output
- Real component of the bus shunt admittance
- Capacitive component of the bus shunt admittance
- Reactive component of the bus shunt admittance

To display the Power System Scaling Dialog, go to the [Tools](#) ribbon tab, and choose **Scale Case** from the [Other Tools](#) ribbon group. When the dialog appears, you may begin to select the buses to be scaled. Buses can be selected individually or in a group by areas or zones. In addition, if you already have specific groups of devices defined as an injection group, you can choose to scale values associated with the injection group.

### Scale by Bus

The button on the left labeled **Bus** enables selection of loads, generators and shunts by the bus to which the devices are attached. Selection of the buses can be done individually, by their area grouping, by their zone grouping, or by their super area grouping. Scaling is done on the buses chosen regardless of the area, zone, or super area specification. Options for areas, zones, and super areas are included only to aid in selecting and unselecting buses to be scaled. This is important to keep in mind if loads or generators are assigned to a different area or zone than their terminal bus.

The **Buses Table** lists the name and number of all buses in the system and whether or not each bus will participate in the scaling. Similarly, the **Areas**, **Zones**, and **Super Area** tables list the names and numbers of all areas, zones or super areas in the system and whether or not each area will participate in the scaling. Simulator initially assumes that no buses are to be scaled. The Power System Scaling Dialog furnishes a number of ways to select the subset of buses to be scaled:

- Use the **Add to Scaling** fields to enter either a [range of areas, zones and/or buses](#) to scale.
- Use the **Remove from Scaling** fields to enter either a [range of areas, zones and/or buses](#) to omit from the scaling.
- Use the **Select buses using a network cut** button to define a [Network Cut](#) to choose a custom set of buses within a portion of the system to be scaled or omitted from the scaling.
- Use the **Buses, Areas, Zones and Super Areas** tables to change the scaling status of individual buses. Simply double-click on the **Scale** field for a bus, area, zone or super area to toggle its value.
- Click the **Set All to Yes** button to scale the entire case. Click the **Set All to No** button to remove the entire case from scaling.

Changing the scaling for an area, zone or super area changes the scaling status for all *buses* in the grouping. For example, to scale all the buses in a single area, first click **Set All to No**. Then, click on the **Scale** field for the desired area in the Area Table. To scale all buses in an area except for a select few, repeat the above process, but then click on the **Scale** field for the buses not to scale.

As you select the buses to be scaled, the fields in the **Totals for Selected Buses** are updated to indicate the total load, generation, or shunt compensation that will be scaled.

Once you have selected the buses, you can either use the **Scale By** fields to enter a new scaling factor for each of the quantities or use the **New Value** fields to specify a new value directly. If you do not wish to scale a particular type of device, such as bus shunts, simply leave the **Scale By** field as unity.

To ensure that the reactive power is scaled proportionately to maintain the current load power factor, click the **Constant P/Q Ratio** option. To enforce generator limits when scaling generation, check the **Enforce Gen MW Limits** option. To scale generation and load to enforce ACE, check **Scale Gen to Keep ACE Constant**. When Simulator scales generation, all generator power outputs at the selected buses are scaled by the specified factor, regardless of area control. To scale only generators whose AGC field is set to *YES*, check **Scale Only AGCable Generation and Load**. To scale both in-service and out-of-service loads, check **Scale Out-Of-Service Loads**.

Finally, click **Do Scaling** to scale the load, generation, or shunt compensation.

### Scale by Area

The button on the left side of the dialog labeled **Area** enables the selection of loads, generators and shunts based on the area designation of the device itself. Loads, generators and shunts can have a different area designation than the terminal bus to which the device is attached. In these cases, it is sometimes necessary to use the **Scale by Area** option to choose only the devices that are designated within a certain area, instead of all devices at a particular bus.

To specify the devices within certain areas to be scaled, toggle the **Scale** property of the areas desired in the Areas table, or use the **Add to Scaling** and **Remove from Scaling** fields to enter [ranges](#) of area numbers to be included or excluded from the scaling.

### Scale by Zone

The button on the left side of the dialog labeled **Zone** enables the selection of loads, generators and shunts based on the zone designation of the device itself. This is identical to the concept described immediately above in the discussion on **Scale by Area**. As was the case with the area designation, loads, generators and shunts can have a different zone designation than the terminal bus to which the devices are attached.

To specify the devices within certain zones to be scaled, toggle the **Scale** property of the zones desired in the Areas table, or use the **Add to Scaling** and **Remove from Scaling** fields to enter [ranges](#) of zone numbers to be included or excluded from the scaling.

### Scale by Injection Group

If injection groups have been defined for the case, you can scale the generation, load, and switched shunt values for the injection group by first clicking on the **Injection Groups** button. The **Buses Table** will be replaced with a new table listing the Injection Groups in the case. To choose the injection groups to scale, double-click the **Scale** field to toggle the value between *No* and *Yes*. As **Scale** fields are toggled to *Yes*, the **Net Injection** fields will update to display the totals for the selected injection groups. When all of the desired injection groups have been selected, either the **Scale By** or the **New Value** fields can be modified for a desired new value for the selected injection groups. Keep in mind that the scaling is done based on net injection and will be done in proportion to the defined [participation points](#).

To ensure that the reactive power is scaled proportionately to maintain the current load power factor, click the **Constant P/Q Ratio** option. To enforce generator limits when scaling generation, check the **Enforce Gen MW Limits** option. When Simulator scales generation, all

generator power outputs at the selected buses are scaled by the specified factor, regardless of area control.

Checking the **Scale in Merit Order** option will scale generators in the order of highest to lowest normalized participation factor. This means that each generator participating in the injection group will be moved to either its maximum or minimum limit, depending on the direction of the scaling, before the output of the next generator in the list is changed, until the total scale amount is achieved. If this option is checked, generator MW limits will be enforced regardless of the status of the **Enforce Gen MW Limits** option.

Check the **Scale Only AGCable Generation and Load** option to scale only those generators and loads whose AGC field is set to *YES*. If the **Ignore AGC flag to calculate participation but use AGC flag to scale individual loads or generators** option is checked, the **Scale Only AGCable Generation and Load** is not enabled and will be ignored. If this option is checked, then all generators and loads, regardless of their AGC status, will be used to calculate the participation percentages for all participating elements. Only those generators and loads whose AGC field is set to *YES* will actually be scaled. Normally, if choosing to scale only AGCable generation and load, only those elements whose AGC status is *YES* will be used to calculate the participation percentages.

The choice of **Scale Starting Point** impacts the reference point from which the scaling starts. **Scale from Present Value** scales by adding an incremental change to the present MW values based on the difference between the **New Value** and **Original Value**. **Scale from Zero** sets all participating elements to zero injection (0 MW load or generation) and then scales to the **New Value** from that starting point. When using the **Scale from Zero** option, only a single injection group can be scaled at a time. The following example explains the difference between these two options:

Suppose that Load 1 is 800 MW and Load 2 is 700 MW. These two loads are the only elements in an injection group and they both have a participation factor of 0.5. The desired **New Value** is 1600 MW. Choosing **Scale from Present Value** will result in Load1 being 850 MW and Load 2 being 750 MW. The incremental change in the total load is 100 MW and both loads pick up half of this change. Choosing **Scale from Zero** will result in Load 1 being 800 MW and Load 2 being 800 MW. Both loads have been scaled to 0 MW initially and they each pick up half of the desired 1600 MW total load.

The **Scale Load Field** options provide a way of scaling load that is out-of-service. Selecting **Actual MW (MW Column)** will scale only those loads that are in-service. (The MW column in the [load case information display](#) reflects the total in-service load at a bus.) Selecting **Modeled MW (ignore load and bus status)** will scale all load regardless of the load and bus status.

Click **Do Scaling** to scale the selected injection groups.

### Scale by Owners

You can scale the generation and load values by ownership by first clicking on the **Owners** button. The **Buses Table** will be replaced with a new table listing the Owners in the case. To choose the owners to scale, double-click the **Scale** field to toggle the value between *No* and *Yes*. As **Scale** fields are toggled to *Yes*, the Generator and Load MW and Mvar fields will update to display the totals for the selected owner. When all of the desired owners have been selected, either the **Scale By** or the **New Value** fields can be modified for a desired new value for the selected owners.

To ensure that the reactive power is scaled proportionately to maintain the current load power factor, click the **Constant P/Q Ratio** option. To enforce generator limits when scaling generation, check the **Enforce Gen MW Limits** option. To scale generation and load to enforce ACE, check **Scale Gen to Keep ACE Constant**. When Simulator scales generation,

all generator power outputs at the selected buses are scaled by the specified factor, regardless of area control. To scale only generators whose AGC field is set to *YES*, check **Scale Only AGCable Generation**. To scale both in-service and out-of-service loads, check **Scale Out-Of-Service Loads**.

Finally, click **Do Scaling** to scale the load and generation for the selected owners.

Note: When scaling generators with multiple owners, the scaling will be done on the entire output of these generators, regardless of the fact that not all of the owners have been set to scale. If this is not a desired behavior, you can define multiple generators at the bus, each with 100% ownership.

## Overview of Facility Analysis in PowerWorld Simulator

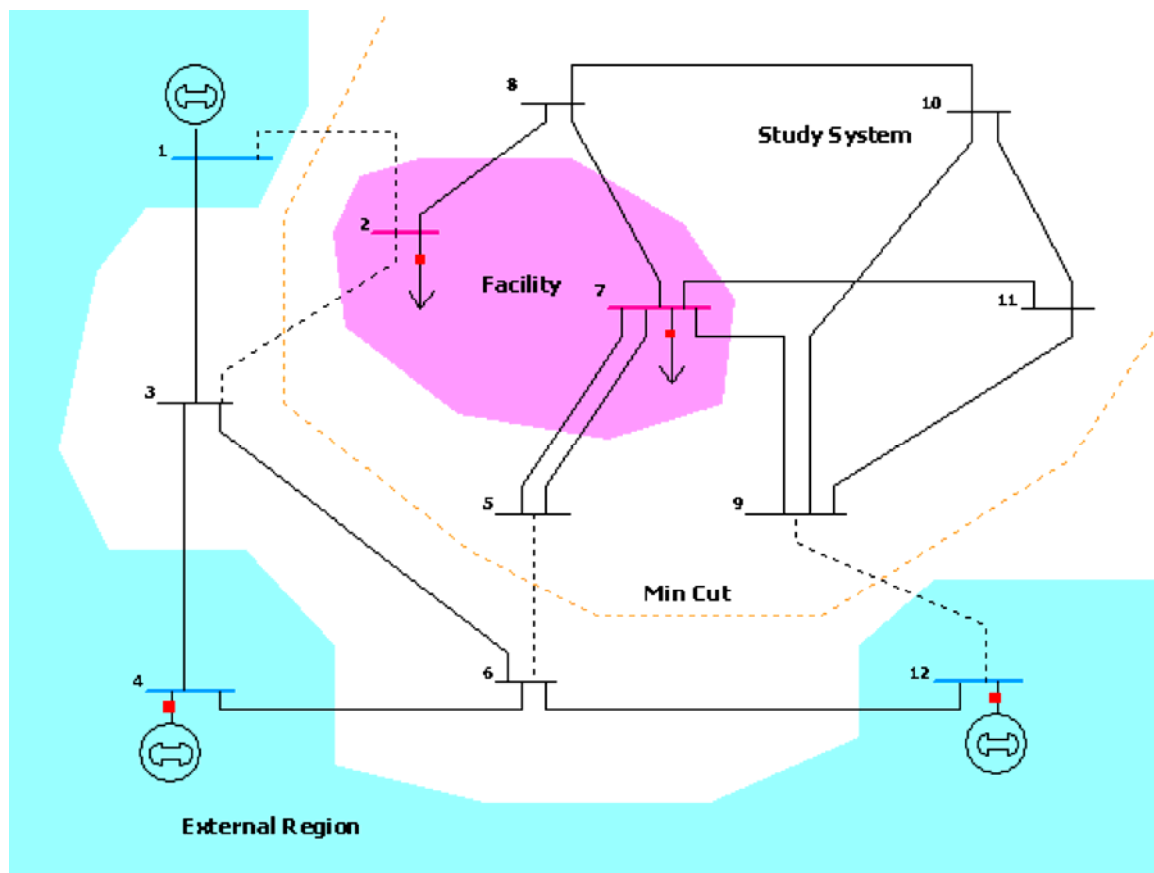
Facility Analysis is used to study the topological redundancy of interconnect-specific electric facilities. This application determines the minimum set of AC transmission lines and transformers that, when opened or removed from the system, would electrically isolate a set of Facility buses from a set of External buses.

The tool is an application of the [augmenting path max flow min cut algorithm](#) with modifications to handle electric networks.

The Facility analysis process has two steps:

1. The [Select the Buses](#) dialog is used to specify the External and Facility buses. Multiple selections of the External buses can be done using any of the area or zone selectors. The Facility buses are specified by setting the **Selected** field of buses to **YES**.
2. The [Facility Analysis dialog](#) is used to determine the Min Cut and visualize the branches that belong to the min cut.

The Facility analysis application runs in Edit Mode and takes into consideration the open or closed status of the branches. Open lines are considered as not present in the system. The figure below illustrates the functionality of the Facility Analysis tool.





---

## Facility Analysis Dialog

---

This dialog is used to determine the branches that would isolate the Facility from the External region as specified in the [Select the Buses](#) dialog. When switching to the Facility Analysis page, the application builds a graph data structure and reports information regarding the External Region and the Facility.

To access the Facility Analysis Dialog go to the [Tools](#) ribbon tab, and choose **Connections > Facility Analysis Dialog** from the [Other Tools](#) ribbon group.

### Select the Buses

The options and use of this page are described in their entirety in the topic titled [Bus Selection Page](#).

### Facility Analysis

#### External Region

The external region is a set of buses from which the Facility would be isolated. Although the buses in the external region may not be adjacent to each other, the algorithm will assume that any of these buses can supply electricity to the Facility and thus the buses in the External region are considered to be connected. The External region is defined using the [Select the Buses](#) dialog.

#### External Region - Number of Buses

Indicates the number of buses in the External region. This number is equal to the number of buses in the system that were labeled as External in the [Select the Buses](#) dialog. There should be at least one bus in the external region.

#### External Region - Capacity

Indicates the number of branches that connect the external buses to study buses in the system. This is the outgoing [graph flow](#) capacity from the External region toward the Facility.

#### Facility

Is the set of buses that constitute the Facility. Although the buses in the Facility may not be adjacent to each other, the algorithm will assume that any of these buses may receive power from the External region, and thus the Facility buses are considered to be connected to each other.

#### Facility - Number of Buses

Indicates the number of buses in the Facility. This number is equal to the number of buses in the system that were labeled as *Study* and whose **Selected** field is set to *YES*. There should be at least one bus in the Facility.

Note that if a bus was specified to be *External* and its **Selected** field is *YES*, then the application will issue an error, since Facility buses cannot be in the External region. In such cases the status will indicate that the graph structure is incomplete.

#### Facility - Capacity

Indicates the number of branches that connect Facility buses with study buses. This number is equal to the outgoing [graph flow](#) capacity from the Facility toward the External region.

#### Status

Shows the status of the Facility Analysis application. If the External region and the Facility are specified, the status will indicate that the graph structure is ready. During execution of the Min Cut algorithm, the status shows the number of the augmenting paths found so far and the number of branches in the current path.

#### **Find Minimum Cut**

Press this button to initiate the [Min Cut algorithm](#). The button is inactive if the graph structure is incomplete, i.e., there is either no external bus or facility bus, or facility buses were found inside the External region.

#### **Show Paths**

Toggle this button to visualize the augmenting paths at the bottom of the form. The augmenting paths are listed in the order in which they were found. Note that the first path has fewer nodes, since the algorithm uses a shortest (least number of nodes) path routine. For each augmenting path the number of buses in the path, as well as the corresponding bus numbers are listed in the dialog. This button is enabled only if a min cut has been found.

---

## Augmenting Path Max Flow Min Cut Algorithm

---

The augmenting path max flow – min cut algorithm is used to identify the minimum number of branches that need to be opened or removed from the system in order to isolate the Facility (power system device) from an External region.

The algorithm is an application of the Max Flow - Min Cut theorem, which states that the maximum flow that can be transferred from a set of source nodes to a set of sink nodes across a graph equals the capacity of the minimum cut. The facility analysis in Simulator finds a min cut, although this cut may not be unique.

The application consists of three stages:

1. **Convert the electric network to a graph structure**  
 In this stage, each branch of the system is converted to an undirected arc with [graph flow](#) capacity equal to one. Thus, only one unit of [graph flow](#) can be sent through a branch. The Facility buses and the External buses are converted to Facility and External supernodes, respectively. This effectively reduces the problem to finding the min cut between these two supernodes. The number of nodes and capacities of the Facility and the External region are also computed during this stage.
2. **Find the Max Flow using the Augmenting Path Algorithm**  
 This is an iterative process. At each step, the algorithm finds a new augmenting path from the Facility to the External region and augments the [graph flow](#) along this path in one unit. Consequently, the branches in the path won't be available for flow augmentation in the next step. Each new path is determined using a shortest path routine. The algorithm stops when no augmenting path from the Facility to the External region can be found. The number of units transferred from the Facility to the External region (path augmentations) reaches the number of branches in a certain cut. Note that the number of branches in the min cut can not exceed the capacity of either the Facility or the External region.
3. **Determine the branches in the min cut**  
 The identification of the branches that constitute the min cut consists in tracking down labels in the buses and branches used during each path augmentation.

---

## Graph Flow

---

Most network and graph theory applications use the concept of flow to represent any object that can be transported, such as communication packets or trucks, but also connectivity properties of graphs. In the [augmenting path max flow min cut algorithm](#) the flow is an artificial concept used to represent topological connectivity of buses. Two buses are adjacent if flow can be sent from one to the other through a branch.

### Graph Flow Capacity of a Branch

Networks that transport some flow are said to be capacitated if its arcs (here synonym of "branches") have some limit associated to the flow transportation. For instance, capacity of a communication channel, or number of trucks that can be simultaneously on a certain road. The algorithm used in the Facility Analysis assigns a capacity of one to each branch. This means that the branch can be used only once for "connecting" two nodes.

### Graph Flow Capacity of a Cut

The capacity of a topological cut is equal to the sum of the capacities of its arcs. In the Facility Analysis, the capacity of the min cut is equal to the number of branches in the min cut, since each branch has a capacity of one.

---

**Chapter 6: Editing Online Diagrams**

---

The Format Menu is used in the Edit Mode to customize the appearance of the online diagrams.

This chapter covers the following:

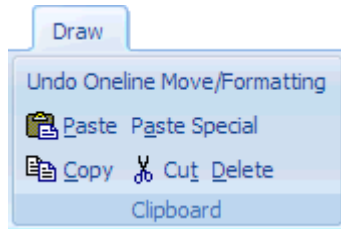
- Editing Overview
- Selecting Objects
- Changing Basic Display Object Properties
- Delete, Copy and Undo

---

## Clipboard Ribbon Group

---

The Clipboard ribbon group contains options for copying and pasting online objects while you are in [Edit Mode](#). It also provides access to the Undo drawing feature. This ribbon group is on the [Draw](#) ribbon tab. The Group is shown below.



### Undo

The Undo command is used in the edit mode to undo the last change made on the online diagram. The Undo command will only undo graphical changes, and will not undo any data changes in the power system model. See [Relationship Between Display Objects and the Power System Model](#) for information on the display/model relationships.

### Paste

The paste command copies the contents of the paste buffer (if any) onto the display at the current cursor location. Use the **Paste** command from the Edit Menu. Note that the paste buffer may contain both display objects and the underlying power system records. When pasting, the display objects are pasted regardless of whether an identical display object already exists on the online. In contrast, duplicate power system records are never pasted. This is because, for example, it is acceptable to have two display objects referring to the same generator, but the generator exists only as a single entity in the power system model. See [Relationship Between Display Objects and the Power System Model](#) for further details.

### Paste Special

Paste Special works the same as Paste, however it will also show a dialog asking the you to choose between

**Absolute Coordinates** : objects will be pasted using the exact same x/y coordinate as the copied objects had. This is useful when copying objects between two different online diagrams which contain the same geographic background.

**Coordinates Relative to Cursor** : objects will be pasted relative to cursor instead of using the exact x/y coordinates as the copied objects.

### Copy

The Copy Command copies the currently selected object(s) into the paste buffer without deleting them. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to copy just the display object(s), or copy both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To copy only the display object(s) and never the power system records from now on, select the **Always Copy Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

## Cut

The Cut Command is used in the Edit Mode to delete the currently selected object(s). To delete a set of objects, first select the objects. Then select the Cut command from the Edit Menu. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to delete just the display object(s), or delete both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To delete only the display object(s) and never the power system records from now on, select the **Always Delete Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Unlike Delete, Cut also copies the selection into the paste buffer.

## Delete

The Delete Command is used in the Edit Mode to delete the currently selected object(s). To delete a set of objects, first select the objects. Then select the Delete command from the **Edit** menu. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to delete just the display object(s), or delete both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To delete only the display object(s) and never the power system records from now on, select the **Always Delete Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Unlike [Cut](#), Delete does not copy the selection into the paste buffer.

---

## Select by Criteria Dialog

---



To open the Select By Criteria Dialog go to the [Draw](#) ribbon tab and choose **Select By Criteria** from the [Select](#) ribbon group.

The Select By Criteria Dialog provides a way of selecting objects that meet a specific set of criteria. The criteria may include Area Numbers, Zone Numbers, Voltage Levels, Zoom Levels, and Object Type.

Use the dialog's controls to specify the selection criteria. Use the **Area** and **Zone** fields to select the areas and zones in which you want to select display objects. [Ranges of area and zone numbers](#) can be entered in the usual way, or you can check the **All Areas** or **All Zones** boxes to select all areas and all zones. You can also use the **Area** and **Zone** tab pages to check or uncheck areas or zones in which you want to select display objects.

Specify the max and min voltage levels for selected objects using the boxes that are provided, keeping in mind that all voltages are in kV.

Specify the [layers](#) for selected objects in the **Layers** tab. If not all layers are selected, the **Layers Range** option will be selected. If the **All Layers** option is clicked, then all the layers will be selected.

Next, select the type of object in which you are interested from the supplied list. To select items from this list, check the box next to the type of field(s) you desire. Display object types that are followed by a right-arrow, such as *Area Fields*, are expandable, meaning that they have several associated subtypes. If you checked a field type that has subtypes, you can highlight the type in the list to see the available subtype list. You can then more specifically select subtypes that you wish to include. By default, all subtypes of a general object type are selected. You also have the option for the list to display all fields available for a particular object type, or to display only the most commonly selected fields.

You can also choose to associate an [advanced filter](#) with a particular type of object in the list for selecting only objects of that type that meet the filter. To do so, right-click in the Filter column next to the object type for which the filter applies, or click on **Add/Modify Filter...** Some objects cannot be associated with an advanced filter, and the option will be disabled when that is the case for an object selected in the dialog.

Checking the option **Group By Object Type** will consolidate all the items by categories. All subtypes will automatically be included as a consequence. The filters are still available.

If the box **Only Show Objects in Display or Already Selected** is checked, only the items or categories (depending on whether the Group By Object Type option is checked) of objects existing or selected in the respective oneline diagram will be shown in the supplied list.

The buttons **Check All** and **Uncheck All** are useful to select/unselect all the items/groups in the list. The button **Check Only Text Fields** is available only when the Group by Object Type is unchecked, and when clicked on all the items that have text associated to them are selected, and the rest of the items are unselected. Clicking on the **Reset To Defaults** button will remove all the current selections and reset them to the default settings.

If you wish to select objects that *do not* have the chosen criteria, then check **Select All Except What Meets the Above Criteria**.

If you wish to only select objects that are currently visible on the diagram (due to layering, etc.), check the box labeled **Select only currently visible objects** to apply the criteria settings only to objects on the diagram that are currently visible.

If the Select by Criteria Dialog is open and some display objects have already been selected, an advanced selection option becomes available. If the box labeled **Use as a filter on**



**presently selected objects** is checked, the criteria chosen in the Select by Criteria Dialog *will only* affect the previously selected objects. In other words, only the objects that were previously selected AND that match the chosen criteria will remain selected when the OK button is pressed. For example, you could use all the options available on the [Select](#) ribbon group on the [Draw](#) ribbon tab to select a group of objects on the oneline. Then using the Select by Criteria Dialog, choose the Bus criteria and check **Use as a filter on presently selected objects** to select only the buses from the rectangular selected group of objects.

Once you have selected the criteria and pressed *OK*, all objects that satisfy ALL the criteria will be chosen. If the Select All Except What Meets the Above Criteria is unchecked, then the selected objects will lie within the specified areas AND zones, the specified voltage level, and the specified zoom levels, and will be of one of the drawing types specified.

You can save your settings for your Select by Criteria session with the case by clicking **Save As** to save with a new name or **Save** to save with the current name. This allows you to re-open the Select by Criteria dialog, and quickly recover settings you may have previously used. Use the **Rename** to rename an already saved set, or **Delete** to remove a previously saved set of criteria.

If you have saved different criteria sets, you can export them all to an [Auxiliary file](#) by choosing **Save to AUX file**. The **Load All from AUX file** will load an entire auxiliary file, regardless of whether it contains Select by Criteria settings or not.

If you wish to clear all settings you have modified on the dialog, click the **Reset to Defaults** button.

Using this dialog in conjunction with the [Format Selection Dialog](#), can be a very fast and easy way to customize your displays.

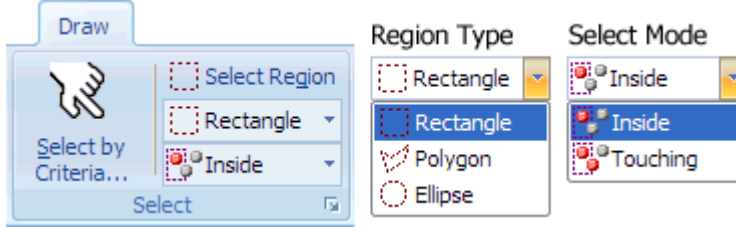
NOTE: When choosing to select all tie lines, all objects associated with the tie line will also be selected. This includes all line fields for the line as well as the two buses that are the endpoints of the line.

---

## Select Ribbon Group

---

The Select ribbon group contains options to help select multiple oneline objects simultaneously. The features in this topic are only available when you are in [Edit Mode](#). This ribbon group is on the [Draw](#) ribbon tab. The Group is shown below.



The features available on this ribbon group are described below.



### Select by Criteria

Click on the Select by Criteria button to open the [Select by Criteria Dialog](#). On oneline diagrams of even a modest size, this is an extremely useful feature and PowerWorld highly encourages you to use this feature.

### Select Region

Use to select all objects in a particular region of the oneline. When the **Region Type** is **Rectangle** or **Ellipse**, then after clicking the button, click and hold the left mouse button on the oneline at the point where you would like to begin the selection. Then, drag the mouse to size the selection shape. Finally, let go of the mouse button once every object you need to select has been selected by the selection rectangle. When the Region Type is **Polygon**, then after clicking this button single click on your oneline diagram for each vertex of the polygon. Finally, finish the selection by double-clicking the mouse after which all objects inside your polygon will be selected.

Depending on the **Select Mode** below, every object located entirely **Inside** the region, or if you have chosen **Touching**, then every object inside or partially touched will be selected. Selected objects will have handles appear indicating that they have been selected.

### Region Type

Specify a **Rectangle**, **Polygon**, or **Ellipse**. This controls the shape used to select objects on the oneline diagram when using the **Select Region** button above.

### Select Mode

This combo box specifies how the **Select Region** button above works. Choose **Inside** in order to select only the objects completely inside the selection rectangle. Choose **Touching** in order to select every object either partially touched by or completely inside the selection rectangle.

There are also other ways to select multiple objects

- You can also select multiple objects manually by holding down the SHIFT key which clicking on objects in Edit Mode. This will also add to your selection.
- If you hold down the SHIFT and CTRL keys simultaneously and then click on your oneline diagram and select a region it has the same affect as click on the **Select Region** button and selecting a region on your oneline

---

## Grid/Highlight Unlinked Objects

---

The Grid/Highlight Unlinked Objects tab is only available in Edit mode. It controls the appearance of a grid in the oneline. A grid is not visible by default, but can be setup from the Grid/Highlight Unlinked Objects page of the [Online Display Options](#) dialog. Additional options are available from this dialog for disabling anchors and highlighting unlinked objects on oneline diagrams.

### Snap Objects to Grid

If checked, new objects placed on the oneline will be aligned with the grid; the grid does not need to be visible on the oneline to align objects to it.

### X Grid Spacing, Y Grid Spacing

These determine the horizontal and vertical spacing of the grid.

### Display Grid Lines on Oneline

If checked, the grid lines will appear on the oneline

### Horizontal Show Every, Vertical Show Every

These numbers determine the density of the visible gridlines; for fewer lines enter higher numbers.

### Gridline Color

Click in the rectangle to choose the color of the gridlines. The default is grey.

### Anchor Options

A check box is available for temporarily disabling anchors on a oneline diagram. This option will disable the anchor properties of all objects on the oneline diagram.

### Do not prompt regarding relinking objects after dragging

Check this check-box to avoid being prompted whether to relink a graphical object after this has been dragged.

## Highlighting of Unlinked Objects

### Highlight Unlinked Objects with Color

Checking this box allows for any objects on all open oneline diagrams that are currently not linked to any data in the case to be highlighted using the highlight color. To change the highlight color used, left-click in the color box to choose a different color.

### Minimum Highlighted Object Pixel Size

The minimum size, in pixels, of the highlight image. This is to prevent the highlight from being unnoticeable when the zoom level is very low.

### Extra Width for Highlighted Lines (pixels)

Sets an extra width parameter, in pixels, for highlighting line objects. This will make line object highlights appear wider than other highlighted objects.

### Display Unlinked Elements in Run Mode

This option allows the highlighting of unlinked elements when you switch to run mode. By default the highlighting of unlinked elements is restricted to edit mode, to avoid confusion in run mode, particularly when drawing [contours](#).

---

### Using the Online Alignment Grid

---

The online alignment grid allows for buses and other objects to be easily aligned when placed onto a online diagram. The alignment grid for a online diagram can be configured in the Online Display Options dialog under the [Grid/Highlight Unlinked](#) page. You can open this display by selecting **Online Display Options** from the [Active](#) ribbon group on the [Onelines](#) ribbon tab.

When the alignment grid is enabled most objects will snap to the grid while being moved on the online. Depending on whether snap to grid is enabled or disabled, objects can individually be made to do the *opposite* of the snap to grid setting by holding down the Alt key while dragging the object.

---

## Setting Background Color

---

The Background Color Dialog changes the background color for the display. To view this dialog, select **Online Display Options** from the [Active](#) ribbon group on the [Onelines](#) ribbon tab, or right-click in the background of the oneline and select **Online Display Options** from the local menu. Switch to the [Display Options](#) tab, click **Change Background Color**, or click on the colored rectangle to bring up the Color Dialog, which you can then use to select the new background color. Click OK to register the new color. The Online Display Options dialog will provide a preview of the color you selected. To define this color as the default background color to use in all new onelines you create, click the **Set as Default Background Color**. Finally, click OK to save your color selection.

---

## Zoom, Pan, and Find

---

The Zoom, Pan, and Find Options Dialog is used to specify a desired zoom level and screen location or to locate a particular bus or area/zone on the oneline. The dialog can be opened by right-clicking on the background of the diagram and choosing **Pan/Zoom Control** from the oneline popup menu, or by clicking on the **Find** button in the [Zoom](#) ribbon group on the [Onelines](#) ribbon tab. The dialog has the following fields:

### Find Objects on Oneline

The Find Object on Oneline section of the dialog is used to pan to any desired [bus object](#), [area/zone object](#), [branch object](#), [interface object](#), or [substation object](#) on the oneline. The dialog makes use of the advanced find functionality of Simulator to locate a device.

#### Only Include Objects Visible at Current Zoom Level

If checked (the default), then the **Object ID** combo box only lists those objects that are visible at the current zoom level.

#### Object Type

Select the type of object to find: *Buses, Areas or Zones, Interfaces, Substations* and *Lines/Transformers*.

#### Sort By

Specify whether you are entering the object by its number or by its name by choosing either **Sort by Name** or **Sort by Number**. This option also determines how the entries in the list of objects are sorted (either by number or by name).

#### Object ID

Enter the object's number or name (depending on the sort type chosen) or select the object from the list.

#### Allow Auto Updating on Selection

Check this box to automatically pan to the specified object. The object will be located at the center of the oneline diagram.

#### Pan to Object on Oneline

Click this button to pan to the specified object if the Auto Updating box is not checked. The object will be located at the center of the oneline diagram.

#### Auto-Zoom when Panning

Check this box to automatically change the zoom when panning to the specified object.

### Zoom / Pan

The Oneline Zoom / Pan tab on this dialog is used to allow the user to specify either a new zoom level and/or screen center and to define these as the new display default settings. The zoom level and screen center can also be changed from the keyboard. Please see [Oneline Zooming and Panning](#) for details.

#### Zoom Level

Enter the desired percentage zoom level (nominal is 100%).

#### Horizontal, Vertical

Enter a desired location for the center of the screen. The nominal screen center is 50 horizontal and 50 vertical.

**Pan/Zoom to New Location**

Changes the screen center and zoom level to the values specified in the above fields.

**Restore Default Values**

Resets the zoom level and screen center to the default values.

---

## Default Drawing Values

---

The [Default Drawing Options Dialog](#) allows you to see and change the various default values used to create new objects. To show this dialog select **Default Drawing Values** from the [Quick Insert](#) ribbon group on the [Draw](#) ribbon tab.



---

## Formatting Ribbon Group

---

The **Formatting** ribbon group contains options for formatting the selected objects on your active online. This group is on the [Draw](#) ribbon tab and its options are available only in Edit mode. The options allow you to control such display object attributes as font, color, line styles, zoom-dependent visibility, and display layer level. The group is shown below.



A description of the actions on this group are as follows.

### Refresh Anchors

Invokes a command to [refresh anchors](#) on all online objects.



### Format Buttons

Clicking any of these five buttons will open the [Format Dialog](#). Each of the buttons opens the Format dialog to a specific tab on the Format Dialog, but they all ultimately open the same dialog.

#### Font

Select this option to change the font with which the selected objects are displayed. The [Font Tab](#) of the Format Selection Dialog is displayed.

#### Line/Fill

Select this option to change the line thickness and style with which a display object is drawn, or the fill color for closed shapes. The [Line/Fill Tab](#) of the Format Selection Dialog is displayed.

#### Levels/Layers

Select this option to change the stack level of an object, the layer the object is contained in, and optional settings for when an object should resize. The [Levels/Layers Tab](#) of the Format Selection Dialog is displayed.

#### Display/Size

Simulator can change the attributes of the way multiple objects are displayed using this option, which opens the [Display/Size page](#) of the Format Selection Dialog. The size of the objects can be adjusted, as well as the orientation of the object(s) from their terminal buses. Objects can also be Anchored to their terminal buses or devices, or marked as Immobile, meaning the object(s) cannot be moved on the online diagram.

#### Field

Select this option to change the attributes related to object text fields which are selected. The [Field Tab](#) of the Format Selection Dialog is displayed.



### Copy Format and Paste Format Buttons

The button on the left is the **Copy Format** button. After selecting an object this will copy all the format properties of that object into an internal buffer. When you then subsequently select other objects, you can click on the button on the right which is the **Paste Format** button. Clicking the **Paste Format** button will bring up the [Paste Format Dialog](#).



### Send to Back, Bring to Front Buttons

After selecting a group of online objects using any of the methods described on the [Select](#) ribbon group, you may then click the **Send to Back** button to cause all the objects selected to move to underneath other objects. Click the **Bring to Front** button to bring objects to the top.

The Send to Back and Bring to Front menu options govern the visibility of display objects that occupy the same screen stack level. All objects have an associated screen stack level, which may be one of Base, Background, Middle, or Top. Objects having a screen stack level of Top obscure all objects having stack levels of Middle, Background, or Base that occupy the same region of the screen. They may or may not obscure objects having the same stack level of Top depending on the order in which the objects having the same stack level are drawn. Selecting Send to Back for a selected object will cause it to be obscured by all other same-level objects that occupy its location. Selecting Bring to Front for a selected object will cause it to be drawn above all other same-level objects that occupy the same region of the screen. The Send to Back and Bring to Front menu options govern relative placement of objects only *within* stack levels. The Send to Back and Bring to Front options do not affect the relative placement of objects having different stack levels.

### Alignment Menu

The alignment menu provides access to the [Alignment Dialog](#) and direct access to options to align the selected objects as specified in the menu items.

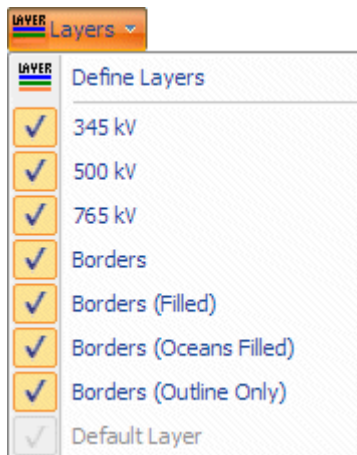
### Grouping Menu

Grouping is only available for background online objects (background lines, ellipses, text, etc...). This menu provides button to Group and Un-Group objects.

### Layers Menu

The Layers Menus provide access for opening the [Define Layers Dialog](#) for the presently active online diagram.

Underneath the button for defining layers is a list of the presently defined layers. You may check or uncheck the various layers to hide or show the objects in those layers.



---

## Format Multiple Objects

---

The Format Selection Dialog features five tabs of controls for modifying the display attributes of selected objects. To open the Format Multiple Options dialog you must first use some of the options available on the [Select](#) ribbon group on the [Draw](#) ribbon tab to select a group of objects. Next, right click on any of the selected objects and select **Format Selection** from the local menu. You may also bring up the Format Multiple Objects dialog by going to the [Formatting](#) ribbon group on the [Draw](#) ribbon tab.

### Line/Fill

The [Line/Fill Tab](#) is used to change the line size, color, and style and the fill color with which the selected objects are drawn.

### Levels/Layers

The [Levels/Layers Tab](#) is used to change the stack level of an object, the layer the object is contained in, and optional settings for when an object should resize.

### Display/Size

The [Display/Size Tab](#) controls the size and orientation of the selected objects.

### Font

The [Font Tab](#) is used to change the font used in text/font objects.

### Field

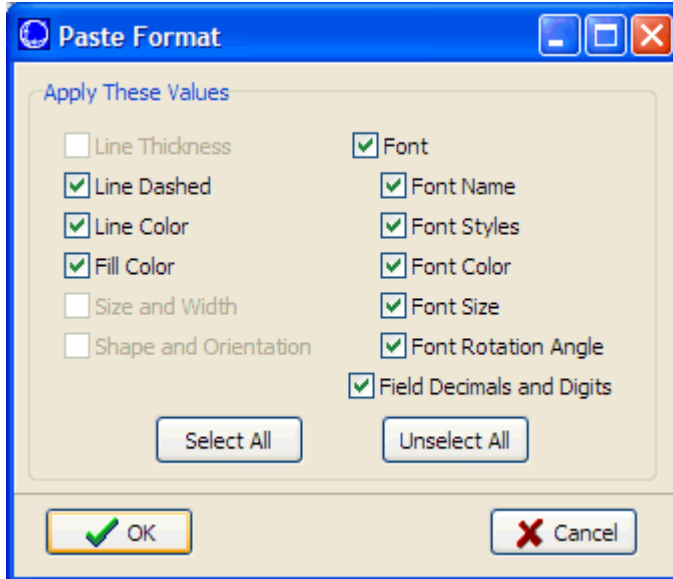
The [Field Tab](#) is used to change selected fields on the diagram to a different field designation. If all of the selected fields are for the same type of object and field, you can select a different field to be represented by the selected objects from the **Field** selection. You can also set the options for including a suffix, the total digits in the field for numeric values, and the number of digits to the right of the decimal.

---

## Paste Format Dialog

---

The Paste Format Dialog appears when you use the Paste Format button on the [Formatting](#) ribbon group on the [Draw](#) ribbon tab. The dialog looks as follows



On this dialog check the attributes you would like to paste to the selected objects. Note that attributes which are disabled in the Paste Format Dialog represent one of two things

- The attribute was not relevant to the objects whose format was copied
- The attribute was not the same for the objects whose format was copied

---

## Font Properties

---


The Font Tab allows you to define the font for the selected objects by choosing Font type, size, color and effects. The page also allows you to control the default display font using the **Make Default** button.

By clicking on **Make Default**, the current font name, size, and effects will be set as the default font for the display. This has the same effect as opening the [Default Drawing Values](#) and setting the default font.

---

## Line/Fill Properties

---

The Line/Fill Tab is used to customize the line size/color and the fill color of selected objects. You can view this tab by selecting  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab. This tab's controls can be used to change the appearance of most, but not all, screen objects. When you modify the line thickness or color of an object, these new values will be set as the default settings for all new line objects. You can view the case defaults using the [Default Drawing Options](#) Dialog. The Line and Fill Options tab has the following fields/commands:

### Line Thickness

Thickness of the line in pixels.

### Dashed

Allows setting a line to appear as a dashed line on the diagram. There are three types of line dashing to choose from in the drop down menu for this option.

**Note:** Due to drawing limitations, the line thickness for a dashed line **must be equal to 1**. Any other line thickness will result in the line being drawn as solid, regardless of the dashed setting.

### Line Color

The Line Color field shows the current line color. To change the line's color, click on the box displaying the current line color, or click the **Change Line Color** button. This displays the Color Dialog. Select the desired color and select OK to change the color, or click Cancel if you do not wish to change the color.

### Line Color 2

The Line Color 2 field applies only to transformer objects. Transformers can be represented by different colors on each side of the transformer coils. This is commonly used to color match each side of the transformer with the color used to represent the voltage level of the transmission system on each particular side of the transformer. However, you can customize the colors for selected transformers by modifying them here.

**Note** that the Line Color 2 field ALWAYS applies to the high voltage side of the transformer when you change

### Use Background Fill

Click on this field to toggle whether or not to fill the background for the selected objects with the selected background fill color. Only text objects such as [Text](#), [Bus Fields](#), [Generator Fields](#), [Load Fields](#), [Switched Shunt Fields](#), [Line Fields](#), [Transformer Fields](#), and [Area Fields](#) and background objects such as [Background Lines](#) and [Background Rectangles](#) can be filled.


### Fill Color, Change Fill Color

The Fill Color field shows the current fill color. To change the fill color, click on the box that displays the current fill color, or click the **Change** button. This displays the Color Dialog. Click on the desired color and the select OK to change the color, or click Cancel if you do not wish to change the color.

---

## Levels/Layers Options

---

The Levels/Layers Tab is used to customize the stack level or layer an object is contained in. To set layers for objects, select the objects on the online diagram (edit mode only) and then choose . The Levels/Layers Options page of the [Format Multiple Objects](#) dialog will be displayed, with the following settings available.

### Stack Level

An object's stack level dictates what objects it will appear above, and which objects it will appear below on a online diagram. For example, circuit breaker and pie chart objects have a default stack level of Top. Therefore anything with a stack level of Middle, Background or Base will appear underneath pie charts and circuit breakers on the online diagram. Objects that are within the same stack level and are drawn in the same location will result in the last object drawn being the visible object on the diagram. You can toggle which elements within the same stack level at the same location is visible using the Send to Back and Bring to Front buttons in the [Formatting](#) ribbon group on the [Draw](#) ribbon tab.

### Layers

Layers are different than stack levels in that layers are designed to allow the user to filter the elements visible on a diagram (in run mode). Initially all diagram objects are in the same layer, called Default. This default layer cannot be modified or deleted. However, you can introduce additional layers using either the **Add New...** or **Define...** buttons. Clicking on **Add New...** will open a dialog to name the new layer, and will automatically add the new layer to the case and change the Layer name in the drop down list to the new layer. Clicking on **Define...** will open the [Screen Layers](#) list display, which will allow you to manage the full list of layers defined for the diagram. Once layers have been defined, you can choose which layer the selected objects belong to by choosing the Layer name from the drop-down list by clicking on the down arrow to the right of the layer name.

### Settings for resizing when zooming

You can modify the maximum and minimum zoom levels at which the selected text fields will no longer resize. Thus when zooming in or out on the diagram, text fields will resize according to the zoom level until the minimum or maximum zoom level are reached. At that point, text fields will no longer resize, but will stay fixed at their current size as you continue to zoom.

### Maintain Fixed Screen Location (do not pan)

This option is available for text fields. When checked for selected text fields, these text fields will no longer pan when you pan the rest of the diagram. This allows you to place text that will always be visible, regardless of what part of the diagram you are observing. You can still move the individual text field itself in edit mode.

### Maintain Fixed Screen Size (do not resize on zoom)

This option is available for text fields. When checked for selected text fields, these text fields will no longer resize when the zoom level of the diagram is changed. This allows you to place text that will always be visible at a constant size, regardless of what zoom level you are currently observing on the online diagram. You can still select the text fields themselves in edit mode and change their font size.

---

## Screen Layers

---

The Screen Layers display can be invoked from either the [Levels/Layers](#) button in the [Formatting](#) ribbon group in Edit Mode, or by right-clicking on the online diagram background and choosing Edit Screen Layers.

Screen layers provide a method to filter objects on a online diagram based on either zoom level or to which layer the objects are assigned. Initially each object is in a common layer called Default, which cannot be modified or deleted. You can create new layers using the Screen Layers display.

Since the Screen Layers display is a [Case Information Display](#), it shares the of other case information displays. You can access many case information display features by right-clicking on the display to invoke the local popup menu. Most importantly, this menu contains the options to Insert and Delete screen layers.

To delete a screen layer, right-click on that layer in the display and select Delete from the popup menu. The one exception is the Default layer, which cannot be deleted.

To insert a new screen layer, right-click on a record in the display and choose Insert from the popup menu. This will open the [Screen Layer Options](#) dialog for defining the options of the new layer you are inserting. You can also define a new layer by going to the [Onelines](#) ribbon tab, then on the [Active](#) ribbon group choose **Screen Layer > Define Screen Layer**.



---

## Screen Layer Options

---

The Screen Layer Options are used to define options for new layers, or modify options for existing layers. Devices can be added to a layer by choosing the layer name from the drop-down list on the [Levels/Layers Tab](#) of the [Format Multiple Objects](#) dialog.

### Name

Layers can have any name desired to describe the layer. The default naming convention is "Layer #".

### Show Layer

If this box is checked, any objects on the diagram which are contained in this layer *may* be visible. This depends on the settings for low and high zoom level, described below. If the box is unchecked, however, then any objects contained in this layer will be hidden. Only objects that are visible will be used when calculating contours. Unlike [Custom Display Detail](#), layers can be applied in both edit mode and run mode.

### Selectable in Edit Mode

By unchecking this box the user prevents the objects contained in this layer of being with the mouse during Edit Mode. This behavior is particularly desirable for elements (such as borders which are pretty much already fixed) when editing online diagrams.

### Use Conditional Display by Zoom Level

Objects can be displayed or hidden based on the Screen Layer to which they are assigned and the current zoom level of the online diagram. Checking this box, and setting the Low Zoom Level and High Zoom Level fields, dictates at what zoom range the objects contained in the zoom level will be visible. If the zoom level is outside the range defined, any objects belonging to the screen layer will be hidden. This also applies to both edit mode and run mode.

### Memo

This tab allows the user to enter text in order to describe this layer.

---

## Format Field Properties

---

The Field tab of the [Format Multiple Objects](#) dialog can be used to modify the format of the selected fields, or to change a set of selected fields from one field representation to another. Use the [Select by Criteria](#) dialog to select fields of a particular type, right-click on any of the selected fields and choose **Format Selection** from the popup menu.

Note that this option also applies to objects which display information related to specific object fields, such as [Line](#) and [Interface](#) Pie Charts.

### Object Type

This field will be filled in automatically based on the type of object the selected field(s) apply to. You cannot change this field, which means that you can only change the selected field(s) to another field for the same type of object.

### Field

The field type of the selected field(s). To change the selected fields to a new field type, click on the drop-down arrow of the combo box, or click on the **Find...** button, and select a new field from the list of fields for the Object Type.

### Include Suffix

Checking this option will add the unit suffix to the selected fields.

### Total Digits in Field

Modify the number of digits to display in the field. This number includes the decimal and the numbers to the right of the decimal.

### Digits to Right of Decimal

Modify the number of the total digits that should appear to the right of the decimal.

---

**Display/Size Properties**

---

Use the Display/Size Tab of the [Format Multiple Objects](#) dialog to change the display size, orientation, anchor property of the selected objects.

**Size**

This field will change the characteristic display size of the selected objects. The characteristic size is normally the vertical size or height of display objects. In objects with different orientation choices, the size is the most distinctive dimension of the object (e.g. in buses with right or left orientations, the size is the horizontal dimension of the bus bar, but in buses with up or down orientations, the size is the vertical dimension of the bus bar).

**Width**

This field will change the characteristic display width of the selected objects. Similarly to the size, the width represents the horizontal dimension or width of display objects. In objects with different orientation choices, the width is the least distinctive dimension of the object (e.g. in buses with right or left orientations, the width is the vertical dimension of the bus bar, but in buses with up or down orientations, the width is the horizontal dimension of the bus bar).

**Anchored**

To toggle whether or not the selected objects are [anchored](#), check or uncheck this option

**Immobile**

To force the selected objects to stay in the same position, check this option.

**Orientation**

This group will modify the orientation setting of the selected objects.

**Shape**

Use this group to modify the shape of the selected display objects. If the option selected is a valid selection for an object, the object shape will be changed; otherwise, the original shape will not be modified. For Substation objects, the Use Substation Layout Settings specifies that the shape selected in the [Substation Display Options](#) will be used.

**Number of Selected Objects**

This field shows the number of objects that are currently selected on the online.

---

## Align Group Objects

---

The Align Group feature allows for selecting multiple objects on a online diagram and aligning them in a specified manner. There are a couple of ways to perform the alignment.

First, select the objects on the diagram that you wish to align. Once you have selected all objects, you can 1) right click on a selected object and choose **Align Group Objects** from the local popup menu or 2) use the **Alignment** menu found in the [Formatting](#) ribbon group on the [Draw](#) ribbon tab.

The **Align Group Objects** option from the local menu will open the Align Dialog, which allows for specifying the type of alignment desired. Alignment can be performed horizontally, vertically, or both. In addition to alignment of the top, bottom, left and right edges of the selected objects, you can also choose to have the objects snap to the [drawing grid](#) in the horizontal and/or vertical directions as well.

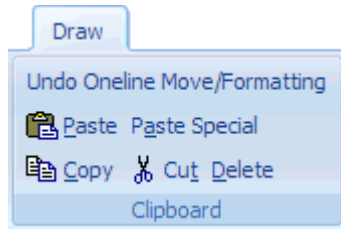
These alignment actions can also be selected directly from the **Alignment** menu found on the Formatting ribbon group on the Draw ribbon tab.

---

## Clipboard Ribbon Group

---

The Clipboard ribbon group contains options for copying and pasting online objects while you are in [Edit Mode](#). It also provides access to the Undo drawing feature. This ribbon group is on the [Draw](#) ribbon tab. The Group is shown below.



### Undo

The Undo command is used in the edit mode to undo the last change made on the online diagram. The Undo command will only undo graphical changes, and will not undo any data changes in the power system model. See [Relationship Between Display Objects and the Power System Model](#) for information on the display/model relationships.

### Paste

The paste command copies the contents of the paste buffer (if any) onto the display at the current cursor location. Use the **Paste** command from the Edit Menu. Note that the paste buffer may contain both display objects and the underlying power system records. When pasting, the display objects are pasted regardless of whether an identical display object already exists on the online. In contrast, duplicate power system records are never pasted. This is because, for example, it is acceptable to have two display objects referring to the same generator, but the generator exists only as a single entity in the power system model. See [Relationship Between Display Objects and the Power System Model](#) for further details.

### Paste Special

Paste Special works the same as Paste, however it will also show a dialog asking the you to choose between

**Absolute Coordinates** : objects will be pasted using the exact same x/y coordinate as the copied objects had. This is useful when copying objects between two different online diagrams which contain the same geographic background.

**Coordinates Relative to Cursor** : objects will be pasted relative to cursor instead of using the exact x/y coordinates as the copied objects.

### Copy

The Copy Command copies the currently selected object(s) into the paste buffer without deleting them. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to copy just the display object(s), or copy both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To copy only the display object(s) and never the power system records from now on, select the **Always Copy Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

## Cut

The Cut Command is used in the Edit Mode to delete the currently selected object(s). To delete a set of objects, first select the objects. Then select the Cut command from the Edit Menu. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to delete just the display object(s), or delete both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To delete only the display object(s) and never the power system records from now on, select the **Always Delete Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Unlike Delete, Cut also copies the selection into the paste buffer.

## Delete

The Delete Command is used in the Edit Mode to delete the currently selected object(s). To delete a set of objects, first select the objects. Then select the Delete command from the **Edit** menu. For power system objects, such as buses, generators or transmission lines, you are given an option of whether to delete just the display object(s), or delete both the display object(s) and their underlying power system records. See [Relationship Between Display Objects and the Power System Model](#) for further explanation of these choices. To delete only the display object(s) and never the power system records from now on, select the **Always Delete Object(s) Only** option. You will not be prompted again. You can disable this selection on the [Default Drawing Options Dialog](#).

Unlike [Cut](#), Delete does not copy the selection into the paste buffer.

---

**Undo Command**

---

The Undo command is used in the edit mode to undo the last change made on the online diagram. The Undo command will only undo graphical changes, and will not undo any data changes in the power system model. See [Relationship Between Display Objects and the Power System Model](#) for information on the display/model relationships.

---

## **Chapter 7: Properties of Simulator Objects**

---

This chapter provides a description of the properties of each of the Oneline Display Objects. Note that when you are using the Simulator, you can obtain context sensitive help for oneline objects by positioning the cursor on the desired object and pressing the F1 key. Of the many properties of the various objects in Simulator, some are relevant only to the Edit Mode, and some only to the Run Mode.

The following material is included:

- Edit Mode Properties and Information
- Run Mode Properties and Information
- General Properties and Information



---

**Zone Information (Edit Mode)**


---

The Zone Dialog is used in the Edit Mode to view information about a zone and to move one or more buses from one zone to another. (See [Zone Information \(Run Mode\)](#) for help on the corresponding Run Mode version.) To view this dialog, open the [Zone Records Display](#) by clicking on **Aggregations > Zones** in the [Model Explorer](#). Then, right-click on the desired zone record and select **Show Dialog** to view this dialog.

The dialog has the following fields:

**Zone Number**

Zone number between 1 and 999. You can use the spin button immediately to the right of this field to move to either the next zone (click the up arrow) or the previous zone (click the down arrow).

**Zone Name**

Alphanumeric identifier for the zone. You can use this field to change the zone's name, provided you click either **Save** or **OK**.

**Find By Number**

To find a zone by its number, type the number in the **Number** field and click this button.

**Find...**

If you do not know the exact zone number you are looking for, you can click this button to open the [advanced search engine](#).

**Labels**

To assign alternative identifying labels to the zone, click the Labels button, which will open the [Label Manager Dialog](#) listing all the labels assigned for the selected zone.

**OK, Save, Cancel**

**OK** saves any changes to the zone name and closes the dialog. **Save** saves any changes to the zone name, but does not close the dialog. **Cancel** closes the dialog ignoring any changes.

The zone dialog contains three additional pages of information:

**Zone Buses Table**

This table lists all of the buses in the zone. Number, name, voltage, area number, and area name are shown for each bus. This table can be used to move buses to a different zone. Select the bus or buses you would like to move with the mouse. Then, enter the **Destination Zone Number**, which is the zone to which you want to move the selected buses. You may enter a zone number that does not already exist, too, so that the buses will be moved to a brand new zone. In this case, be sure to provide the **Zone Name**, as well. Finally, click the **Move Selected Bus(es) to Destination Zone** button to implement the move.

**Custom**

The Custom Page of the Zone Information dialog displays custom numbers or strings defined with the viewed zone. The [Memo](#) box is simply a location to log information about the zone. To log information about the zone, simply start typing your information or comments about the zone in the memo box.

---

**Bus Information (Edit Mode)**


---

This dialog is used to view/modify information about each bus in the system during Edit Mode. It is very similar in content to its [Run Mode](#) counterpart.

**Bus Number**

Unique number between 1 and 2,147,483,647 (equals  $2^{31}$  minus 1) used to identify the bus. You can use the spin button immediately to the right of the number to move to the next bus (click the up arrow) or the previous bus (click the down arrow).

**Find By Number**

To find a bus by its number, enter the number into the **Bus Number** field and then click this button.

**Bus Name**

Unique alphabetic identifier for the bus.

**Find By Name**

To find a bus by its name, enter the bus name into the Bus Name field (case insensitive) and then click this button.

**Find...**

If you do not know the exact bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

**Nominal Voltage**

Nominal voltage of the bus in kV.

**Labels**

Clicking on this button will open the [Label Manager Dialog](#) listing all of the labels assigned for the selected bus.

**Area Number**

Number of the bus' area. Each bus must be associated with an area record. If a bus is specified as belonging to an area that does not already exist in the model, the new area is created.

**Area Name**

Alphabetic identifier for the bus' area. If the area already exists, you do not need to enter this value.

**Zone Number**

Number of the bus' zone, between 1 and 2,147,483,647 (equals  $2^{31}$  minus 1). Each bus must be associated with a zone record. If a bus is specified as belonging to a zone that does not already exist in the model, the new zone is created. Zones provide a useful mechanism for breaking up a large system. Buses can be assigned to zones independent of their area assignments. Thus, a single area could contain multiple zones, or a single zone could span multiple areas. You can use the [Zone Dialog](#) to list the buses in a particular zone and easily move a group of buses from one zone to another.

**Zone Name**

Alphabetic identifier for the bus' zone. If the zone already exists, you do not need to enter this value.

**Owner Number**

The number of the bus' owner.

**Owner Name**

The name of the bus' owner.

**Substation Number**

The number of the substation the bus is contained in. This is typically blank in most power flow data, unless specifically added to a PowerWorld binary save file.

**Substation Name**

The name of the substation the bus is contained in. This is typically blank in most power flow data, unless specifically added to a PowerWorld binary save file.

**OK, Save, and Cancel**

**OK** saves your changes and closes the dialog. **Save** saves your changes but does not close the dialog; this allows you to use, for example, the **Find By Number** command to edit additional buses. **Cancel** closes the dialog without saving any changes.

### Bus Information

**Voltage (pu), Angle (degrees)**

Current per-unit voltage magnitude and angle for the bus. If you are inserting a new bus into an existing system, **you should not change the initial per unit voltage values**. Rather, when you first switch to Run Mode, Simulator will estimate the voltage magnitude and angle at the new buses in such a way as to reduce the initial mismatches. **This automatic estimation is only available if you have not modified the voltage in any way.**

**System Slack Bus**

Check only if the bus should be modeled as a system slack bus. Each case requires at least one slack bus.

### Display

**Orientation**

Set the orientation of the bus. The current choices are right, left, up or down. For the current bus object shapes of Rectangle and Ellipse, right and left are analogous to horizontal, up and down are analogous to vertical. Additional shapes for buses may be available in the future for which right, left, up and down may have more specific impact on the appearance of the bus object.

**Shape**

Sets the shape of the bus object.

**Width**

Specifies the horizontal axis of an elliptical bus or horizontal side length of a rectangular bus.

**Size**

Specifies the vertical axis of an elliptical bus or the vertical side length of a rectangular bus.

**Scale Width with Size**

When this option is checked, changing the size will cause the width to automatically adjust to keep the same ratio of width to size. To adjust the width independent of the size, uncheck this option or adjust the width separately after adjusting the size.

#### **Link to New Bus**

If you have right clicked on a bus and opened the bus information dialog, you could change the bus number in the **Bus Number** field and press this button to force the bus object on the diagram to link to the new bus number and information in the load flow data.

### **Attached Devices**

#### **Load Summary Information**

Displays the net MW and Mvar load at the bus. You cannot change either of these fields from this display. Select the Add or Edit Bus Load Records to view the individual load records for the bus via the [Load Dialog](#).

#### **Shunt Admittance**

The real (G) and reactive (B) components of shunt compensation at the bus, expressed in MW and MVR, respectively.

### **Fault Parameters**

This tab is only visible when viewing the bus information for a bus with attached load. The parameters on this tab are used when running a fault analysis study. The values represent the total load at the bus for the negative and zero sequence as equivalent admittances. By default, these values are zero. For load buses, these values can be changed by the user, or they can be specified by loading short circuit data from within the [Fault Analysis Dialog](#).

### **Custom**

The Custom page of the dialog contains two sections: custom fields and memo.

The custom fields section allows access to setting and changing the values for custom fields that have been defined for the bus. Defining custom fields is detailed in [Custom Field Descriptions](#).

The [Memo section](#) of the dialog is simply a location to log information about the bus. Any information entered in the memo box will be stored with the case when the case is saved to a [PWB](#) file.

---

## Bus Field Information

---

Bus field objects are used primarily to indicate various quantities associated with bus devices. Furthermore, some bus field types, which are distinguished by an integrated spin button, may be used to change bus device properties.

The Bus Fields Information Dialog can be used to modify the properties of individual bus fields on the oneline. The dialog displays the following fields:

### Find...

If you do not know the exact bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

### Bus Number

Number of the bus associated with the field. Use the drop-down box to view a list of all buses in the case with [valid area/zone/owner filters](#).

### Bus Name

Name of the bus associated with the field. Use the drop-down box to view a list of all buses in the case with [valid area/zone/owner filters](#).

### Total Digits in Field

Total number of digits to show in the field.

### Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

### Field Value

The current value of the field being displayed.

### Field Prefix

A prefix that can be specified and displayed with the selected value.

### Delta Per Mouse Click

This option is currently not used for bus fields.

### Maintain Constant Load Power Factor

This option is only used for certain fields, such as load fields and generator fields. It is not used currently for any bus fields.

### Include Suffix

If this checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

### Anchored

If this checkbox is checked, the bus field is [anchored](#) to its associated bus, which means that it will move with the bus.

### Rotation Angle in Degrees

The angle at which the text will appear on the diagram.

### Type of Field

Used to determine the type of bus field to show. The following choices are available:

**Bus Name** Name of the bus

**Bus Number** Bus number (from 1 to 2,147,483,647 (equals  $2^{31}$  minus 1))

**Bus Voltage** Bus voltage magnitude

**Bus Angle** Bus voltage angle in degrees

**MW Marginal Cost** Bus MW marginal cost in \$/MW•hr; available only with OPF

**Mvar Marginal Cost** Bus Mvar marginal cost in \$/Mvar•hr; available only with OPF

**MW Loss Sensitivity** Increase in losses for 1 MW of generation over the nominal

**Select a Field** Choose from any defined Bus fields

Select **OK** to save changes and to close the dialog, or click **Cancel** to close the dialog without saving your changes.

---

## Shortest Path Between Elements

---

To access the Shortest Path Between Elements go to the [Tools](#) ribbon tab, and choose **Connections > Determine Shortest Path Between...** from the [Other Tools](#) ribbon group. The Determine Shortest Path Between dialog provides a way to find the shortest electrical pathway between two parts of the power system.

After clicking the **Calculate** button, the shortest path of AC branches will be determined with the first bus being inside the Start Element and the last bus being inside the End Element. Options described below allow the user to choose the Start Element, End Element, Distance Measure for each branch, and branches that are allowed to be traversed. The results will be displayed in a case information display at the bottom of the dialog showing the sequence of buses that are traversed between the start and end elements.

The options on the dialog are described below.

### Start Element Type and Element

Choose an element type to be either a Bus, Substation, Area, Zone, Super Area or Injection Group. Then use the object chooser to choose the particular object to be the starting element. For more help on the object choose see the [Find Dialog Basics](#).

### End Element Type and Element

Choose an element type to be either a Bus, Substation, Area, Zone, Super Area or Injection Group. Then use the object chooser to choose the particular object to be the ending element. For more help on the object choose see the [Find Dialog Basics](#).

### Distance Measure

Choose the distance measure that will be used to determine distances between nodes. Each branch will be treated as having a length based on the choice below. Note: negative values are not allowed, therefore negative values will be treated as extremely small lengths instead.

X	the per unit series reactance.
Z	the magnitude of the series impedance (based on the per unit series reactance and resistance)
Length	the Length Field for each branch
Number of Nodes	Length of 1.0 is used for all branches
Other	When choosing Other, click the Find.. button to choose any numeric field of a branch

### Lines to Process

Regardless of the Distance Measure above, you can choose which branches allowed to be traversed when finding the shortest path.

All	All branches are allowed to be traversed
Only Closed	Only branches that are presently closed can be traversed
Filter	Only branches that meet the <a href="#">Advanced Filter</a> specified can be traversed . Click <b>Define...</b> to choose or create an advanced filter.
Selected	Only branches whose Selected? field is set to YES can be traversed



---

**Substation Information (Edit Mode)**


---

The Substation Dialog is used in the [Edit Mode](#) to view information about a substation and to move one or more buses from one substation to another. (See [Substation Information \(Run Mode\)](#) for help on the corresponding [Run Mode](#) version.) To view this dialog, first select **Aggregation > Substations** from the [Model Explorer](#) to view the [Substations Records](#) Display. Then, right-click on the desired substation record and select **Show Dialog** to view this dialog.

The dialog has the following fields:

**Substation Number**

An integer identifier for the substation. You can use the spin button immediately to the right of this field to move to either the next substation (click the up arrow) or the previous substation (click the down arrow).

**Substation Name and ID**

Two alphanumeric identifiers for the substation. You can use these fields to change the substation's name or ID, provided you click either Save or OK.

**Find By Number**

To find a substation by its number, enter the number into the **Substation Number** field, then click this button.

**Find By Name**

To find a substation by its name, enter the name into the **Substation Name** field, then click this button.

**Find By Sub ID**

To find a substation by its substation ID, enter the ID into the **Substation ID** field, then click this button.

**Find...**

If the exact substation number, name and ID are not known, you can use the [Find Dialog](#) to search for and select a substation from a list of substations.

**Labels**

Clicking this button will open the [Subscribed Aliases dialog](#) displaying the list of defined labels for the substation. New labels can also be added for the substation from the dialog as well.

**Buses**

This table lists all of the buses in the substation. Number, name, voltage, area number and name, and zone number and name are shown for each bus. This table can be used to move buses to a different substation. Select the bus or buses you would like to move with the mouse. Then, enter the Destination Substation Number, which is the substation to which you want to move the selected buses. You may enter a substation number that does not already exist, too, so that the buses will be moved to a brand new substation. In this case, be sure to provide the new substation a name and ID, as well. Finally, click the Move Selected Bus(s) to Destination Substation button to implement the move.

**Display Options**

The Display Options tab of the Substation Information dialog is only visible if an open diagram contains an object representing the substation. This tab allows you to choose the general appearance of the substation object. Use the Width and Height fields to set the **width** and **height** of the substation object. The **Shape** field allows you to choose what shape the substation object will take.

By clicking on **Link to New Object**, the Choose Object selector will appear, allowing the user to select another substation in the load flow data to be linked to this particular graphical object.

The **Substation Layout Online, URL or Command** field allows you to specify the same of the Simulator oneline diagram (pwd) file that should be automatically opened if you click on the substation object on the diagram in run mode. If no oneline is specified, Simulator will search for a default name of the format AreaName\_SubstationName. If a diagram cannot be found, then Simulator will not attempt to open any oneline diagrams when the substation is clicked. Optionally, a HTTP URL can be entered in the form <http://example.com>. Also, a line command can be entered, such as *Excel.exe c:\file.xls*.

#### **Substation Generators, Loads, and Switched Shunts Tables**

These tables list all of the generators, loads, and switched shunts in the substation. Number, name, ID, status, and additional fields for each type of device are shown.

#### **Custom**

Enter any text notes you wish in the Memo page. When the case is saved as a Simulator PWB file, the memo text will also be saved. Custom fields can also be entered for storage with the selected substation.

#### **Geography**

Displays geographic information about the location of the substation, in Latitude and Longitude.

#### **OK, Save, Cancel**

OK saves any changes to the substation name or ID, and closes the dialog.

Save saves any changes to the substation name or ID, but does not close the dialog.

Cancel closes the dialog ignoring any changes.

---

## Substation Field Options

---

Substation field objects are used to show different values associated with substations. This dialog is used to view and modify the parameters associated with these fields.

### Substation Number

Select the number of the substation for which you are inserting or viewing information of a substation field.

### Find...

If you do not know the exact substation you are looking for, you can click this button to open the [advanced search engine](#).

### Substation Name

The name of the currently selected substation.

### Substation ID

The substation ID number.

### Total Digits in Field

Total number of digits to show in the field.

### Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

### Rotation Angle in Degrees

The angle at which the text will appear on the diagram.

### Field Value

Shows the current output for the super area field. Whenever you change the **Type of Field** selection, this field is updated.

### Field Prefix

A prefix that can be specified and displayed with the selected value.

### Anchored

Toggle the anchoring of the field to the substation object of which it is associated.

### Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

### Type of Field

Used to determine the type of super area field to show. The following choices are available:

#### Substation Name, Substation Number

Name or number of the selected substation.

#### Substation ID

ID string for the selected substation.

#### Max Nominal Voltage

Displays the nominal voltage of the highest nominal voltage bus in the substation.

#### Substation Load MW, Substation Load Mvar

Total Load MW or MVAR in the substation.

**Substation Gen MW, Substation Gen MVAR**

Total Generator MW or MVAR in the substation.

**Select a Field**

Select this option and click **Find Field...** to display the value of any other substation specific field.

---

## Generator Information (Edit Mode)

---

This dialog is used to view and modify the parameters associated with each generator in the system. It can also be used to insert new generators and sometimes to delete existing generators.

The Edit Mode version of the Generator Information Dialog is almost identical to the [Run Mode](#) version.

### Bus Number

Unique number between 1 and 2,147,483,647 (equals  $2^{31}$  minus 1) used to identify the bus to which the generator is attached. The dropdown list enumerates all generator buses in the case that meet the criteria established by [area/zone/owner filters](#). You may select a bus number directly from the dropdown list, or you may use the spin buttons to cycle through the list of generator buses.

### Bus Name

Unique alphabetic identifier for the bus to which the generator is attached, consisting of up to eight characters. Use this dropdown box to view a list of all generator bus names in the case with valid [area/zone/owner filters](#).

### ID

Two-character alphanumeric ID used to distinguish multiple generators at a bus; '1' by default.

### Find By Number

To find a generator by its number and ID, enter the number into the **Bus Number** field and the ID into the **ID** field. Then click this button.

### Find By Name

To find a bus by its name and ID, enter the bus name into the **Bus Name** field (case insensitive) and the ID into the **ID** field. Then click this button.

### Find...

If you do not know the exact generator bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

### Status

Status of the generator, either Closed (connected to terminal bus) or Open (not connected). You can use this field to change the status of the generator.

### Area Name

Alphabetic identifier for the terminal bus' area.

### Labels

Clicking on this button will open the [Subscribed Aliases dialog](#) listing all the labels or aliases assigned for the selected generator.

### Fuel Type

Type of fuel used by the generator this model represents. In most cases, this field is unnecessary for normal load flow analysis, and hence the default value is Unknown. However, this value can be useful during the [Security Constrained OPF](#) analysis.

### Unit Type

The type of unit the generator represents, such as combined cycle, steam, hydro, etc.

There are six additional areas of information on this dialog for specific aspects of generation:

[Display Information](#)

[Power and Voltage Control](#)

[Costs](#)

[Fault Parameters](#)

[Owner, Area, Zone, Sub](#)

[Custom](#)

---

## Generator Field Information

---

Generator field objects are used primarily to indicate various quantities associated with generation devices. Furthermore, some generator field types, which are distinguished by an integrated spin button, may be used to change generation device properties.

The Generator Fields Information Dialog can be used to modify the properties of individual generator fields on the oneline. The dialog displays the following fields:

### Find...

If you do not know the exact bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

### Bus Number

Number of the bus to which the generator associated with the field is connected. Use the dropdown box to view a list of all buses with generators in the case with [valid area/zone/owner filters](#).

### Bus Name

Name of the bus to which the generator associated with the field is connected. Use the dropdown box to view a list of all buses with generators in the case with [valid area/zone/owner filters](#).

### ID

ID of the generator associated with the field. Generator ID are two-character alphanumeric fields.

### Total Digits in Field

Total number of digits to show in the field.

### Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

### Field Value

The current value of the field being displayed.

### Field Prefix

A prefix that can be specified and displayed with the selected value.

### Delta Per Mouse Click

Generator fields can be used not only to show various fields associated with generation devices, but they can also be used to change some values. This is accomplished using spin buttons shown to the right of the generator field. When the up spin button is clicked, the generator field value is increased by the amount specified in the *delta per mouse click* field. When the down spin button is clicked, the generator field value is decreased by the same amount.

This field is only used for fields of the following types: Setpoint Gen MW, Setpoint Gen Mvar, and Gen Setpoint Voltage. Specifying a nonzero value in this field causes the integrated spin button to appear as part of the generator field on the oneline.

### Rotation Angle in Degrees

The rotation angle at which the text field should be displayed.

### **Anchored**

If the *Anchored* checkbox is checked, the generator field is [anchored](#) to its associated generator, which means that it will move with the generator.

### **Include Suffix**

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

### **Type of Field**

Used to determine the type of generator field to show. The following choices are available:

**Gen MW Output** MW generation

**Gen Mvar Output** Mvar generation

**Gen AGC Status** AGC status of generator; status can be toggled in Simulator

**Gen AVR Status** AVR status of generator; status can be toggled in Simulator

**Gen Setpoint Voltage** Voltage of generator on bus

**Select a Field** Choose from any of the different generator fields

Select **OK** to save changes and to close the dialog, or click **Cancel** to close the dialog without saving your changes.



---

**Generator Options: Display**

---

This information is located on the [Generator Information Dialog](#).

**Display Size**

The size of the generator.

**Scale Width with Size**

If checked, the Display Width will automatically be scaled to the appropriate setting when the Display Size is changed. If unchecked, then only the length of the generator object will be affected by changing the value of the Display Size.

**Display Width**

The width of the display object. This setting is automatically set if Scale Width with Size is checked and the value of the Display Size field is changed, or the Display Width value can be set manually to a new value.

**Pixel Thickness**

Thickness of the display object in pixels.

**Orientation**

Specifies the direction in which to draw the object.

**Anchored**

If checked, the object is anchored to its terminal bus. See [Anchored Objects](#) for details.

**Link to New Generator**

Links the object to a different generator in the data.

---

## Generator Options: Power and Voltage Control

---

This information is located on the [Generator Information Dialog](#).

### Power Control

The Power Control grouping fields are used to show/change the values associated with the real power output of the generator.

#### MW Output

Current real power output of the generator.

#### Minimum and Maximum MW Output

Minimum and maximum real power output limits for the generator. Simulator will not let the MW output go below its minimum value or above its maximum value if the *Enforce MW Limits* option is exercised.

#### Available for AGC

Determines whether or not the generator is available for automatic generation control (AGC). Normally this box should be checked. However, there are times when you would like to control the generator output manually (such as if you are using the generator to remove a line limit violation), in which case you should leave this box unchecked. A generator is also placed on "manual" control anytime you manually change its output. You could then place the generator back on AGC control if you wish it to participate in an area generation dispatch.

#### Enforce MW Limits

If checked, the minimum and maximum MW limits are enforced for the generator, provided the **Enforce Generator MW Limits** field is also checked on the Limits Tab of the [Simulator Options Dialog](#). If this box is checked and a generator is violating a real power limit, the generator's MW output is immediately changed.

#### Participation Factor

The participation factor is used to determine how the real power output of the generator changes in response to demand when the generator is available for AGC and the area is on [participation factor control](#). When you open a case using the [PTI Raw Data Format](#), this field is initialized to the per unit MVA rating of the generator, since participation factor information is not stored in the PTI format.

### Voltage Control

The Voltage Control grouping is used to show/change values associated with controlling the voltage/reactive power output of the generator.

#### Mvar Output

Current reactive power output of the generator. You can manually change this value only if **Available for AVR** is not checked.

#### Min and Max Mvar Output

Specify the minimum and maximum allowable reactive power output of the generator.

#### Available for AVR

Designates whether or not the generator is available for automatic voltage regulation (AVR). When the AVR field is checked, the generator will automatically change its reactive power output to maintain the desired terminal voltage within the specified reactive power range. If a reactive limit is reached, the generator will no longer be able to maintain its voltage at the setpoint value, and its reactive power will then be held constant at the limit value.

#### Use Capability Curve

If checked, then the generator's reactive power limits are specified using a reactive capability curve that prescribes the dependence of the generator's reactive power limits on its real power output. Otherwise, the fixed values given in the **Min Mvar Output** and **Max Mvar Output** fields are used. The generator reactive capability can be defined using the table that appears at the bottom of the dialog. Please see [Generator Reactive Power Capability Curve](#) for details.

#### Regulated Bus Number

Number of the bus whose voltage the generator is regulating. This is usually, but not always, the generator's terminal bus. Multiple generators can regulate the same remote bus, but the regulated bus must not be another generator bus. If the generator is at a slack bus, it must regulate its own terminal voltage. Select **Solution Details > Remotely Regulated Buses** in the [Model Explorer](#) to view the [Remotely Regulated Buses](#) display, which identifies all buses that are being remotely regulated.

#### SetPoint Voltage

Specifies the desired per unit setpoint voltage value the generator is to regulate at the regulated bus. The regulated bus need not be the generator terminal bus.

#### Remote Reg %

This field is only used when a number of generators at different buses are regulating a remote bus (i.e., not their terminal buses). This field then specifies the percentage of the total reactive power required by the remote bus to maintain its voltage that should be supplied by this generator. The default value is 100. If the total value is different from 100%, then all the regulation factors are normalized to obtain a percentage of regulation.

---

## Generator Options: Costs

---

The **Costs** tab of the Generator Information dialog (run mode) is used to show/change values associated with the cost of operation of the generator. See [Generator Cost Information](#) for details. Cost data can also be saved/loaded using the [Generator Cost Data files](#).

### Output Cost Model

#### Cost Model

Simulator can model generators as not having a cost model, or having either a cubic cost model or a piecewise linear model. The cost model type you choose determines the content of the remainder of this dialog

#### Unit Fuel Cost

The cost of fuel in \$/MBtu. This value can be specified only when you have chosen to use a cubic cost model.

#### Variable O&M

The Operations and Maintenance costs. Only used for cubic cost models.

#### Fixed Costs (costs at zero MW Output)

The fixed costs associated with operating the unit. These costs are independent of the generator's MW output level and is added to the cost prescribed by the cubic or piecewise linear model to obtain the total cost of operating the generator in \$/MWHr. The total fixed cost is the addition of the fuel cost independent value and the fuel cost dependent value multiplied by unit fuel cost.

#### Cubic Cost Coefficients A, B, C, D

For cubic cost models of the form  $C(P_{gi}) = (d \cdot P_{gi}^3 + c \cdot P_{gi}^2 + b \cdot P_{gi}) \cdot (\text{fuel cost}) + \text{fixed costs}$  (as described above), specify the cost curve's coefficients. The A coefficient is historically the fuel cost dependent fixed cost value, which is combined separately now with the fuel cost independent value. These coefficients can be specified only when you have chosen to use a cubic cost model.

#### Piecewise Linear Table

If you have chosen to use a piecewise linear cost model, a table appears that allows you to specify pairs of MW output levels and corresponding generator operating costs. To insert a new point on the cost curve, right-click on the table and choose *Insert New Point* from the resulting local menu. To delete an existing point from the cost curve, right-click on the table and choose *Delete Point* from the resulting local menu. To edit an existing point in the table, simply enter your changes to the appropriate cells.

#### Convert Cubic Cost to Linear

Use this option to create a piecewise linear cost function from the cubic cost function specified by the coefficients A, B, C, and D and the fuel cost. Specify the number of break points, and hence the number of segments, in the **Number of Break Points** field. Click the **Convert to Linear Cost** button to create the piecewise linear function that approximates the cubic cost function. This action switches **Cost Model** option to *Piecewise Linear* and displays the **Piecewise Linear Table** that identifies the piecewise linear curve's breakpoints.

### Bid Scale / Shift

#### Cost Shift, Cost Multiplier

The cost shift and cost multiplier allow you to easily apply a shift to the cost function for the purpose of assessing how variations in bids impact profit. The cost function is affected based on the following equation:

$$(\text{Original Cost Function} + \text{Cost Shift}) * \text{Cost Multiplier}$$

**Marginal Cost (run mode only)**

Shows the marginal cost of producing real power at the generator at its current output level,  $dC_i(P_{gi})/dP_{gi}$ .

**ED/OPF Cost (run mode only)**

This is the cost of production for this generator following an economic dispatch or optimal power flow solution, *including* the scaling from the cost shift and cost multiplier fields.

**Unscaled Cost (run mode only)**

The cost of production of the generator, *ignoring* the cost multiplier and cost shift. This cost is the result of the original cost function by itself.

---

## Generator Options: Fault Parameters

---

The parameters on this tab are used when running a [fault analysis](#) study.

### Generator MVA Base

The assumed MVA base for the generator. This value is used when calculating fault analysis values for the internal generator parameters.

### Neutral Grounded

Check this check-box if the generator has the neutral grounded.

### Generator Step Transformer

The resistance, reactance and tap setting for the generator step-up transformer, if one is being modeled internally with the generator. By default, no internal transformer model is assumed.

### Internal Impedance

These fields represent the internal impedance of the generator for all three sequences. By default, all three values are initially the same as the load flow internal impedance of the generator. All three sets of values can be modified, either manually or by loading values from an external file using the [Fault Analysis Dialog](#).

### Neutral-to-Ground Impedance

Neutral-to-ground impedance for the generator. These values get implemented with the zero sequence admittance matrix. Note that the neutral-to-ground impedance will not be used, even if specified, if the original model for the generator implicitly models the generator step-up transformer. This is because the implicitly modeled transformer is assumed to have a delta winding on the generator side of the transformer, which isolates the generator from the rest of the zero sequence network.

---

**Generator Options: Owners, Area, Zone, Sub**

---

This information is located on the [Generator Information Dialog](#).

This tab is used to display or change the generator's owner information, area information, zone information, and substation information.

**Same Owner as Terminal Bus**

Read-only check-box that indicates whether the generator's owner is the same than the terminal buses' owner.

**Owners**

Currently, Simulator supports up to four owners for generators. To add an owner of a generator, change one of the Owner fields to a new owner number, and update the owner percentages accordingly. To modify an owner's percentage of ownership, simply modify the value in the percentage field for that owner. If you set the percentage of an owner to 0, that owner will be removed from the list of owners for the device. You can also remove an owner from owning part of a device by changing the owner field for that owner to 0. Note that if you do not set the new owner percentages of all specified owners such that the total is 100%, Simulator will normalize the percentages such that the total is 100% when you click **Save** or **OK** on the generator dialog.

**Area Number, Area Name**

The area number and name to which the generator belongs. Note that you can change the area of the generator to be different than the area of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the generator within a different area than that of the bus to which it is electrically connected.

**Zone Number, Zone Name**

The zone number and name to which the generator belongs. Note that you can change the zone of the generator to be different than the zone of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the generator within a different zone than that of the bus to which it is electrically connected.

**Substation Number, Substation Name**

The name and number of the substation to which the generator belongs, and it is the same than that of the terminal bus.

---

**Generator Cost Description**


---

The cost associated with operating a generator varies according to the output of the generator, with the general rule that getting more power out of a generator costs more. In Simulator, there are two options for modeling generator cost. The first employs the common cubic relationship

$$C_i(P_{gi}) = \text{IndFixedCost} + (a_i + b_i P_{gi} + c_i (P_{gi})^2 + d_i (P_{gi})^3) * \text{fuel cost } \$/\text{Hour}$$

where  $P_{gi}$  is the output of the generator at bus  $i$  in MW. IndFixedCost is the fuel cost independent fixed cost, and therefore is not multiplied by the fuel cost. The value  $a_i$  is the fuel cost dependent fixed cost, and is therefore multiplied by the fuel cost. The values  $b_i$ ,  $c_i$ , and  $d_i$  are used to model the generator's input-output (I/O) curve. The I/O curve specifies the relationship between how much heat must be input to the generator (expressed in MBtu per hour) and its resulting MW output. Normally, the cubic coefficients remain constant for a generator. The last term in the equation is the fuel cost, expressed in \$/MBtu. This value varies depending on the fuel used in a generator. Typical values would be \$ 1.25/MBtu for coal and \$ 2/MBtu for natural gas. The resultant equation is known as the fuel-cost curve. The value of the fixed costs,  $b_i$ ,  $c_i$ ,  $d_i$ , and fuel cost can be viewed and modified using the [Generator Information Dialog](#).

Simulator can also model generator costs using a piecewise linear model consisting of pairs of MW output and incremental cost (\$/MWhr) of generation, along with a fixed cost. These piecewise linear curves must be convex curves, meaning the marginal cost of the current MW break point must be higher than the previous MW break point.

Such curves can be defined using the [Generator Information Dialog](#) or by loading data from [generator cost data](#) files.



---

## Set Generator Participation Factors

---

Participation factor control is another of Simulator's mechanisms for distributing an area's responsibility to serve its load, losses, and interchange. It is particularly well-suited to implementing [automatic generation control \(AGC\)](#) when you do not have good economic information for an area's generators. With participation factor control, the amount of power that each generator contributes to meeting its areas load, loss, and interchange responsibilities is controlled by the size of its participation factor. The unit that has the largest participation factor contributes the most, and the unit that has the smallest participation factor contributes the least.

The **Set Generator Participation Factors Dialog** gives you a convenient way to define the participation factors for multiple generators. You can set the participation factor according to a number of different formulae and then apply this prescription to all generators in a specific area, all generators in a specific zone, all generators in the system, or all generators whose display filters are currently set to true.

To display the **Set Generator Participation Factors Dialog**, you first need to open the [Area Information Dialog](#) and switch to the Area MW Control Options page. The Area Information Dialog has a button labeled **Set Participation Factors** that is enabled only if the Participation Factor Control option is selected under the Area Control Options heading. Set the area on participation factor control by selecting the Participation Factors option, and then press the **Set Participation Factors** option.

The Set Generator Participation Factors Dialog is divided into two parts. The first part, which occupies the top half of the form, allows you to indicate how the participation factors should be calculated or set for each generator. Your options include:

**Max MW Rating of Generator** The participation factor for each generator is set to the generator's maximum MW capability.

**Difference Between Max and Current Output** The participation factor for each generator is set to the generator's reserve power, so that each generator participates in proportion to how much it has left to contribute.

**Constant Value of** The participation factor for each generator is set to the same hard-coded value.

**File** The participation factor for each generator is read from a file. The first line of the file should contain the keyword NUMBERS or NAMES indicating whether generators are identified by bus number or by bus name in the file. All subsequent lines should be comma-delimited and contain three fields: the number or name of the generator's bus, the generator's id, and the generator's participation factor.

If you choose any of the first three options, you then must tell Simulator to what generators you want to assign the participation factors. To assign the participation factors to all generators in a specific area, select the **All Generators in Area** option, and then choose the area from the adjacent dropdown box. If you want to assign the participation factors to all generators in a specific zone, select the **All Generators in Zone** option, and then choose the zone from the adjacent dropdown box. If you want to assign the participation factor to all generators in the system regardless of their area or zone affiliation, select the **All Generators in System** option. Finally, if you want to assign the participation factor to just those generators whose display filter criteria evaluates to true, choose the **All Generators With Valid Display Filters** option.

If you instead chose to read participation factors from a file, only those generators whose factors you read from the file will have their factors set by this action. However, unless each generator's associated area is set to control generator output using participation factor control, this information will be ignored. To make sure that each generator's area is set to participation factor control, check the **Set Corresponding Areas to Participation Factor Control** box. Then, each corresponding area will be set to participation factor control.

---

## Generator Reactive Power Capability Curve

---

The reactive power output of most generators depends on the real power output of the generator. This dependence is expressed using a reactive capability curve. Simulator models the reactive capability curve using a piecewise linear approximation. The reactive capability curve is modified on the [generator dialog](#) and can be saved/loaded using the [Generator Reactive Capability Curve Auxiliary files](#).

### Modeling a Reactive Power Capability Curve From the Generator Dialog

- Make sure the **Use Capability Curve** checkbox is checked.
- In the table immediately below the checkbox, prescribe the reactive capability curve using up to 50 points. The points should be ordered by MW, in numerically increasing order. At each point specify the MW value, the minimum reactive power value in Mvar, and the maximum reactive power value in Mvar. The first point should correspond to the minimum MW output of the generator, while the last point should correspond to the maximum MW output.
- To insert a new point, click on the desired column, and then right-click to display the table's local menu. Select Insert Point.
- To remove a point, click on the desired column, and then right-click to display the table's local menu. Select Delete Point.
- When finished be sure to select **Save** to save your modifications.

In the Run Mode, you can view the reactive power capability curve graphically by right-clicking on the generator to display its submenu and then selecting **Reactive Capability Curve**.

---

**Load Information (Edit Mode)**


---

This dialog is used to view and modify the parameters associated with each load in the system. It can also be used to insert new loads and sometimes to delete existing loads. It is nearly identical in structure to its [Run Mode](#) counterpart.

The Load Information Dialog can be used to inspect and modify the model of a bus load. To view the Load Information Dialog, select the load and choose **Show Dialog** from the [case information toolbar](#) or right-click on the load of interest and select **Load Information Dialog** from the resulting local menu. The dialog has the following fields:

**Bus Number**

Unique number between 1 and 2,147,483,647 (equals  $2^{31}$  minus 1) used to identify the bus to which the load is attached. The dropdown box provides a list of all load buses with valid [area/zone/owner filters](#). You can use the spin button to cycle through the list of load buses.

When you insert objects graphically, the Bus Number and Bus Name fields are usually set automatically to the bus upon which you placed the object.

**Bus Name**

Unique alphabetic identifier for the bus to which the load is attached, consisting of up to eight characters. The dropdown box lists the names of all load buses in the case with valid [area/zone/owner filters](#).

**ID**

Two-character ID used to distinguish multiple loads at a bus. By default, the load ID is equal to "1 ." An identifier of '99' is used to indicate an equivalent load.

**Find By Number**

To find a load by its number and ID, enter the number into the Bus Number field and the ID into the ID field. Then click this button.

**Find By Name**

To find a load by its name and ID, enter the bus name into the Bus Name field (case insensitive) and the ID into the ID field. Then click this button.

**Find...**

If you do not know the exact load bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

**Status**

Status of the load, either Closed (connected to terminal bus) or Open (not connected). You can use this status field to change the load's status.

**Labels**

Clicking on this button will open the [Label Manager](#) dialog listing all the labels assigned for the selected load.

**Area Number, Area Name**

Number and name of the area the load is a member of.

**Zone Number, Zone Name**

Number and name of the zone the load is a member of.

**Substation Number, Substation Name**

Number and name of the substation the load and the terminal bus are members of.

**Owner Number, Owner Name**

Number and name of the owner the load is a member of. If the load owner is the same as the terminal bus owner, the **Same Owner as Terminal Bus** box will be checked. Loads DO NOT have to be owned by the same owner as the terminal bus.

**OK, Save, Delete, and Cancel**

**OK** saves your changes and closes the dialog. **Save** saves your changes but does not close the dialog; this allows you to use, for example, the Find By Number command to edit additional loads. **Delete** deletes the current load. **Cancel** closes the dialog without saving your changes.

Specific load information can be found on the following pages of the Load Options dialog:

[Load Information](#)

[OPF Load Dispatch](#)

[Custom](#)

---

## Load Options: Load Information

---

This page of the Load Options dialog contains information on the load magnitude and display settings.

### **MW and Mvar Value Fields**

The MW and Mvar Value fields are used to represent the amount of base real and reactive load at the bus. Usually this load is modeled as being "constant power," meaning that the amount of load is independent of the bus voltage magnitude. However, Simulator also permits modeling "constant current" load, for which the load varies in proportion to the bus voltage magnitude, and "constant impedance" load, for which the load varies in proportion to the square of the bus voltage magnitude. Values in these fields are specified in MW and MVR assuming one per unit voltage. All six fields can be modified by the user.

### **Display Size**

Size of the load.

### **Scale Width with Size**

If checked, the Display Width will automatically be scaled to the appropriate setting when the Display Size is changed. If unchecked, then only the length of the generator object will be affected by changing the value of the Display Size.

### **Display Width**

The width of the display object. This setting is automatically set if Scale Width with Size is checked and the value of the Display Size field is changed, or the Display Width value can be set manually to a new value.

### **Pixel Thickness**

Thickness of the display object in pixels.

### **Orientation**

Specifies the direction to draw the object.

### **Anchored**

If checked, the object is anchored to its terminal bus. See [Anchored Objects](#) for details.

### **Link to New Load**

Links the object to a different load record in the data.

---

**Load Options: OPF Load Dispatch**

---

This tab of the Load Options dialog contains settings for allowing the load to be included as an OPF Control. The load(s) can then be dispatched in the OPF algorithm according to the assigned costs.

**Benefit Model**

If this field is set to none, the load will not be dispatchable in the OPF solution. If the option is set to Piecewise Linear, the load is dispatchable during the OPF, according to the following fields.

**Min. and Max. MW Output**

Minimum and maximum load MW demand for OPF dispatch.

**Available for AGC**

If checked, the load will be available for redispatch during the OPF routine.

**Fixed Benefit**

Value of the load benefit at minimum demand.

**Piece-wise Linear Benefit Curve**

This table allows you to specify pairs of MW demand levels and corresponding load benefit values, which in turn define the starting points and slopes of the piece-wise linear benefit curve segments. To insert a new point on the cost curve, right-click on the table and choose *Insert New Point* from the resulting local menu. To delete an existing point from the cost curve, right-click on the table and choose *Delete Point* from the resulting local menu. To edit an existing point in the table, simply enter your changes to the appropriate cells.

---

## Load Field Information

---

Load field objects are used primarily to indicate various quantities associated with load devices. Furthermore, some load field types, which are distinguished by an integrated spin button, may be used to change load device properties.

The Load Fields Information Dialog can be used to modify the properties of individual load fields on the oneline. The dialog displays the following fields:

### Find...

If you do not know the exact bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

### Bus Number

Number of the bus to which the load associated with the field is connected. Use the dropdown box to view a list of all buses with loads in the case with [valid area/zone/owner filters](#).

### Bus Name

Name of the bus to which the load associated with the field is connected. Use the dropdown box to view a list of all buses with loads in the case with [valid area/zone/owner filters](#).

### ID

ID of the load associated with the field. Load ID fields are two characters in length.

### Total Digits in Field

Total number of digits to show in the field.

### Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

### Field Value

The current value of the field being displayed.

### Field Prefix

A prefix that can be specified and displayed with the selected value.

### Delta Per Mouse Click

Load fields can be used not only to show various fields associated with load devices, but they can also be used to change some values. This is accomplished using spin buttons shown to the right of the load field. When the up spin button is clicked, the load field value is increased by the amount specified in the *delta per mouse click* field. When the down spin button is clicked, the load field value is decreased by the same amount.

This field is only used for fields of the following types: Load MW and Load Mvar. Specifying a nonzero value in this field causes the integrated spin button to appear as part of the load field on the oneline.

Note that the **Maintain Constant Load Power Factor** option will allow you to specify a Delta per Mouse-click for the MW load, and when the MW value is changed in run mode, the MVAR load will also change in such a way as to keep the power factor of the load constant.

### Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.



**Anchored**

If the *Anchored* checkbox is checked, the load field is [anchored](#) to its associated load, which means that it will move with the load.

**Rotation Angle in Degrees**

The angle at which the text is placed on the diagram, in degrees.

**Type of Field**

Used to determine the type of load field to show. The following choices are available:

**Load MW** MW load

**Load Mvar** Mvar load

**Select a Field** Choose from any of the different load fields

Select **OK** to save changes and to close the dialog, or click **Cancel** to close the dialog without saving your changes.

---

**Load Modeling**


---

Each load can be modeled as having voltage variation. The voltage variation is modeled using the Base Load Model fields on the [Load Information](#) dialog. Thus the actual real and reactive value of each load is determined using the following equation:

$$MW = \text{Load MW Multiplier} * (SMW + IMW * V + ZMW * V * V)$$

$$Mvar = \text{Load Mvar Multiplier} * (SMvar + IMvar * V + ZMvar * V * V)$$

where

MW current real power load in MW

Mvar current reactive power load in Mvar

SMW constant power MW value

Smvar constant power Mvar value

IMW constant current MW value (assuming 1.0 per unit voltage)

Imvar constant current Mvar value (assuming 1.0 per unit voltage)

ZMW constant impedance MW value (assuming 1.0 per unit voltage)

Zmvar constant impedance Mvar value (assuming 1.0 per unit voltage)

V per unit bus voltage magnitude

Load MW Multiplier = (Area MW Multiplier) \* (Zone MW Multiplier)

Load Mvar Multiplier = (Area Mvar Multiplier) \* (Zone Mvar Multiplier)

The Area Multipliers scale all of the loads in an area. The Zone Multipliers scale all of the loads in a zone. These are static values that do not vary over time. If a time varying schedule of loads is required, the [Time Step Simulation](#) tool should be used instead of the static multipliers.

---

## Line/Transformer Information (Edit Mode)

---

The Line/Transformer Information Dialog is used to view and modify the parameters associated with each transmission line and transformer in the system. You can also insert new transmission lines and delete existing transmission lines from this dialog.

The Edit Mode version of this dialog is very similar in content to its [Run Mode](#) counterpart.

The Line/Transformer Information Dialog has the following fields:

### From Bus Number and Name

*From Bus* number and name. For transformers, the *from bus* is the tapped side.

### To Bus Number and Name

*To Bus* number and name.

### Circuit

Two-character identifier used to distinguish between multiple lines joining the same two buses. Default is '1'.

### Find By Numbers

To find a line or transformer by its bus numbers, enter the *from* and *to* bus numbers and the circuit identifier. Then click this button. Use the spin button to cycle through the list of lines and transformers in the system.

### Find By Names

To find a line or transformer by the names of its terminal buses, enter the *from* and *to* bus names and the circuit identifier. Then click this button.

### Find...

If you do not know the exact from and to bus numbers or names you are looking for, you can click this button to open the [advanced search engine](#).

### From End Metered

This field is only used for lines and transformers that serve as tie lines, which are lines that join two areas. If this field is checked for a tie line, then the *from* end of the device is designated as the metered end. Otherwise the *to* end is metered. By default, the *from* end is metered. The location of the metered end is important in dealing with energy transactions because it determines which party must account for transmission losses.

### Default Owner (Same as From Bus)

Read-only check-box that indicates whether the line's owner is the same than the from bus' owner.

### From and To Bus Area Name

Control area name in which each terminal bus is located.

### From and To Bus Nominal kV

Nominal voltage level of each terminal bus, in kV.

### Labels

Clicking on this button will open the [Subscribed Aliases dialog](#) listing all the labels or aliases assigned for the selected branch.

### OK, Save, Delete, and Cancel

**OK** saves your changes and closes the dialog. **Save** saves your changes but does not close the dialog; this allows you to use, for example, the *Find By Number* command to edit additional transmission lines. **Delete** deletes the current transmission line. **Cancel** closes the dialog but does not save any changes.

### Display

See [Transmission Line/Transformer Display Options](#).

### Parameters

See [Transmission Line/Transformer Parameters Options](#).

### Transformer Control

This tab is only visible for transformer objects. See [Transformer Modeling](#) for details on modeling either LTC or phase shifting transformers.

### Series Capacitor

This tab is only visible for series capacitor objects. See [Series Capacitor](#) for details on modeling series capacitors.

### Fault Info

The parameters on this tab are used when running a [fault analysis](#) study. The values represent the zero sequence impedance and zero sequence line shunt admittances for the analysis. By default, the positive and negative sequence line impedances and line shunt admittances are the same as the load flow impedance. The same fields are used for transformers, along with the configuration field. The configuration field defines the winding type combinations for the transformer (wye, delta, etc.) As a default, Simulator assumes a grounded wye to grounded wye transformer, which has the same model as a transmission line. Usually transformers are not of this type, and the proper type would need to be defined either manually or loaded from an external file in order for the fault analysis to be accurate.

### Owner, Area, Zone, Sub

The **Default Owner (Same as From Bus)** read-only check-box indicates whether the line's owner is the same than the from bus' owner. Transmission elements can have up to four different owners, each with a certain owner percentage. To add an owner of a transmission element, change one of the Owner fields to a new owner number, and update the owner percentages accordingly. Note that if you do not set the new owner percentages of all specified owners such that the total is 100%, Simulator will normalize the percentages such that the total is 100% when you click **Save** or **OK** on the line/transformer dialog.

The area, zone and substation to which the From and To buses belong, are also shown.

### Custom

This page of the dialog can be used to enter notes about the transmission line. Any information entered in the memo box will be stored with the case when the case is saved to a PWB file. Custom fields can also be populated with user-specified values and will be saved with the case as well.

---

**Transmission Line/Transformer Options: Display**

---

**Pixel Thickness**

Thickness of the display object in pixels.

**Anchored**

If checked, the object is anchored to its terminal bus, which means that it will move when you move the terminal bus. See [Anchored Objects](#) for details.

**Link to New Line**

Use the *Link to New Line* button to create a new line corresponding to the entries you have made in the dialog. This button performs the same function as pressing *Save*. Note that adding a new line to the case in this way does not add a transmission line to the oneline display; the new line is present only in the model. You may then add the newly modeled line to the oneline diagram in the usual way (such as select **Network > Transmission Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab).

**Symbol Segment**

Only visible for transformer and series capacitor objects. This field specifies which "segment" of the branch contains the transformer or capacitor symbol. A segment constitutes a section of the line between vertex points of the line object, and are numbered starting at the from bus.

**Symbol Size**

Only visible for transformer and series capacitor objects. Specifies the size (width) of the transformer or capacitor symbol on the branch.

**Symbol Percent Length**

The distance or "length" of the symbol on the segment of the branch which contains it.

**Show Detailed Line Vertices**

When checking this option, a new group of information appears, showing the x,y coordinates of every vertex of the line. Additionally to being able to edit those locations, it is possible to shift or scale all the values with the buttons **Shift All Values** and **Scale All Values**, respectively.

---

## Transmission Line/Transformer Options: Parameters

---

### Status

Current status of the device.

### Length

If the length of the line is known, it can be entered here for informational purposes.

### Per Unit Impedance Parameters

The series resistance and reactance, the total charging susceptance (that is,  $B$ , not  $B/2$ ) and the shunt conductance of the device (in per unit). In the case of transformers, the magnetizing conductance and susceptance can also be specified.

### Line Shunts

Select to view the [Line Shunt](#) Dialog. This dialog is used to only view or modify the values of the line shunts in Run Mode. Line shunts are expressed in terms of the per-unit conductance and susceptance at each end of the line or transformer. If the line has shunts, the check box **Has Line Shunts** is checked.

### MVA Limits

Ratings for the transmission line or transformer in MVA. Simulator allows the use of up to eight different limits.

### Calculate Impedances

This will display a pop-up menu with the following two options:

#### ...From Per Distance Impedances

Clicking this item will open the [Line Per Unit Impedance Calculator dialog](#), which can be used to convert actual impedance and current limits to per unit impedance and MVA limits, and vice versa.

#### ...From Conductor Type and Tower Configuration

Clicking this item will open the [Transmission Line Parameter Calculator](#) dialog, which can be used to compute per unit impedance values, based on a conductor type and a tower configuration.

### Convert Line to Transformer

Clicking this button turns the currently selected transmission line into a transformer, making the transformer specific fields available.

---

## Line Shunts Information

---

The Line Shunts Information Dialog is used to modify the parameters of transmission line shunts. The modeling of the line shunts is explained below.

### Line Shunts

Line shunts are included in the model as admittance-to-ground values at either the From end or the To end of a transmission line. These values can represent many things, such as shunt-to-ground capacitors, reactors, zigzag (or grounding) transformers, or equivalenced system values. Line shunt values are entered in per unit, with a positive B corresponding to capacitors and a negative B corresponding to reactors. Mathematically, the line shunts are included in the algorithm in the same manner that line charging capacitance is included using the Pi model.

Line shunt values can be modified from the [Transmission Line Information](#) dialog.

---

## Line Per Unit Impedance Calculator Dialog

---

The Line Per Unit Impedance Calculator Dialog allows to convert actual impedance and current limit values to per unit impedance and MVA limit values, and vice versa.

The dialog has the following elements:

### Actual Impedance and Current Limits

This part of the dialog shows all the impedance related values in Ohms/length-unit as well as the transmission line limits specified in Amps. If any of these values is modified, the corresponding per unit or MVA value will be changed accordingly, taking into consideration the current line length, length units, and system base values.

### Line Length

This value indicates the length of the line in miles or kilometers, depending on the units selected in **Length Units**. The option **When changing convert** is used to convert the values when the length value is modified. The option **PU/MVA -->** indicates that the actual impedance and current limits will be converted to per unit impedance and MVA limits when the length value is changes. The option **<-- Electrical** specifies that the per unit impedance and MVA limits will be converted to actual impedance and current limits when the length value is modified.

### Length Units

This option indicates the length units. Choices are miles and kilometers. When this parameter is changed, the user will be prompted to confirm to convert the actual impedance and current limits, as well as the line length values, from the old units to the new units. If the answer is positive, then the actual impedance and current limits, and the line length will be the same but they will be expressed in the new units selected. If the answer is negative, the values will not change numerically but they still will be expressed in the new units selected.

### System Base Values

The system base values show the power base, the voltage base, and the impedance base. These values can not be modified in this dialog.

### Per Unit Impedance and MVA Limits

This part of the dialog shows all the impedance related values in per unit as well as the transmission line limits specified in MVA. If any of these values is modified, the corresponding actual or Amps value will be changed accordingly, taking into consideration the current line length, length units, and system base values.

### Conductor Type

Shows the conductor type selected to compute the per unit impedances in the [Transmission Line Parameter Calculator](#) dialog. It will be blank if there isn't any.

### Tower Configuration

Shows the tower configuration selected to compute the per unit impedances in the [Transmission Line Parameter Calculator](#) dialog. It will be blank if there isn't any.

### Calculate PU Impedances From Conductor Type and Tower Configuration

Clicking this button will open the [Transmission Line Parameter Calculator](#) form.



---

## Transmission Line Parameter Calculator

---

The transmission line parameter calculator is a tool designated to compute characteristic line parameters give the type of the conductor and the configuration of a three-phase overhead transmission line.

The parameters computed are the resistance R, reactance X, susceptance B, and conductance G. These values are computed as distributed (per unit of distance), lumped or total (for a specific line distance), and in per-unit.

### Calculations

The following controls are part of the calculations section:

#### Parameters Calculation

This section is to enter the necessary data to compute the characteristic line parameters that are shown in the Results panel.

#### Input Data

**Conductor Type:** This is the combo box that lists all the conductor types available in the Conductors table. To add, remove or edit a specific conductor and its characteristics, see Conductor Type section below.

**Tower Configuration:** This combo box lists all the tower configurations available in the Tower Configurations table. To add, remove or edit a specific tower configuration, please go to the Tower Configuration section below.

**Line Length:** This is the value of the distance of the transmission line. The units are miles when using English system, or kilometers when using the Metric (SI) system.

**Line Length Units:** The line length units specify the measurement system to use when entering the line length. The options are English system or Metric (SI) system. The final and intermediate results will also be shown in the system specified here.

**Power Base:** The system voltampere base in MVA.

**Voltage Base:** The line-line voltage base in KV.

**Impedance Base:** The impedance base in Ohms. This value is automatically computed when the power base and the voltage base are entered or modified.

**Admittance Base:** The admittance base in Siemens. It is also automatically computed as the inverse of the impedance base.

#### Final Results

When all the input data is entered, the results automatically will be displayed. The values for R, X, B and G are shown in three different sections, each section corresponding to Distributed, Lumped or Total, and Per Unit values, respectively. The Power Surge Impedance Loading is calculated only for the lumped section.

#### Intermediate Results

The intermediate values calculated to compute the R, X, B, and G values are displayed here. The computed geometric mean radius and geometric mean distance are shown in the Distributed values section. The characteristic impedance and propagation factor are displayed in the Lumped values section.

Note: To see the specific calculations used in this program, see the Calculations section, at the end of this document.

### Ampere to MVA Limit Conversion

This section converts the limits of the transmission line from Amperes to MVAs, given the voltage base, and vice versa.

## Conductor Type

This section is used to add, remove, rename, and edit the information related to the conductor types. This can be done in two ways: using the form for an individual conductor type, or using the table for all the conductor types available.

### Edit By Form

Conductors are identified by a unique code word. All the available conductors are listed in the Conductor Code Word combo box. By selecting one conductor, all its properties are displayed in the form. There, the user is able to modify any of those values. After modification of any value, the user has to save the changes by clicking on the button **Save** before changing tabs, otherwise the changes will be lost.

By clicking on **New**, a message prompting for a name for a new conductor will be shown. By clicking on **Rename**, a new name for the current conductor type is required. In order to save the current conductor type with a different name is necessary to click on **Save As**. Finally, to remove the current conductor the user can do so by clicking on the **Delete** button.

### Edit By Table

In this tab, all the conductor types are shown as records in a table, where every field is a characteristic of the conductor. In order to edit the records in this table, use the Database button described in the Database section. While editing the table, the user can not change of tab until changes are posted or discarded.

### Conductor Properties

The available properties of the conductors are the following:

Code Word: Code name for the type of conductor. The names of bird species are typically used. Code Word has to be unique.

Area: The area of aluminum conductor in circular mils. A circular mil is the cross-sectional area of a circular conductor having a diameter of 1 mil. One mil is one thousandth of an inch (0.001").

Aluminum strands: Number of aluminum strands.

Steel layers: Number of steel strands.

Aluminum layers: Number of aluminum layers.

External diameter: Outside diameter of the conductor in inches.

GMR: Geometric Mean Radius in feet.

DC Resistance: DC resistance of the conductor at 20°C per 1 mile in Ohms.

AC Resistance 25: AC resistance of the conductor at 60 Hz and 25°C per 1 mile in Ohms.

AC Resistance 50: AC resistance of the conductor at 60 Hz and 50°C per 1 mile in Ohms.

AC Resistance 75: AC resistance of the conductor at 60 Hz and 75°C per 1 mile in Ohms.

Inductive Reactance: Inductive reactance per conductor at 1 foot spacing at 60 Hz in Ohms/mile.

Capacitive Reactance: Capacitive reactance per conductor at 1 foot spacing at 60 Hz in MegaOhms/mile.

### Tower Configuration

This part is used to add, remove, rename, and edit the information related to the tower configurations. This can be done in two ways: using the form for an individual tower configuration, or using the table for all the tower configurations available.

#### Edit By Form

Tower configurations are identified by a unique name. All the available tower configurations are listed in the Tower Configuration Name combo box. By selecting one specific tower configuration, all its characteristics are displayed in the form. There, the user can modify any of those characteristics. After modification of any value, the user has to save the changes by clicking on the button **Save** before changing tabs, otherwise the changes will be lost.

By clicking on **New**, a message prompting for a name for a new tower configuration will be shown. By clicking on **Rename**, a new name for the current tower configuration is required. In order to save the current tower configuration with a different name is necessary to click on **Save As**. Finally, to remove the current tower configuration the user can do so by clicking on the **Delete** button.

#### Edit By Table

In this tab, all the tower configurations are shown as records in a table, where every field is a value of the tower configuration. In order to edit the records in this table, use the Database button described in the Database section. While editing the table, the user can not change of tab until changes are posted or discarded.

#### Tower Configuration Values

The values of the tower configuration are the following:

Name: Name for the tower configuration. Name has to be unique.

Phase spacing: x and y coordinates of phases A, B and C positions. Values are in feet for English system and meters for Metric (SI) system. When these values are modified, the phase spacing graph is automatically updated. **Draw axis** has to be checked for x and y axis to be drawn in the graph.

Conductors per bundle: This section specifies the number of conductors (either single conductor or a bundle of conductors) per phase. The maximum number of conductors per bundle allowed is 4.

Use regular spacing: When using a bundle of conductors, checking the **Use Regular Spacing of** check box will use the specified value as a regular spacing among the conductors. If the **Use Regular Spacing of** check box is not checked, then the custom conductors spacing section has to be used.

Conductors spacing: x and y coordinates of the conductors in the bundle. Values are in feet for English system and meters for Metric (SI) system. When these values are modified, the bundle spacing graph is automatically updated. **Draw axis** has to be checked for x and y axis to be drawn in the graph.

Temperature: Assumed temperature in Fahrenheit degrees for English system and Celsius degrees for Metric (SI) system.

Frequency: Frequency of the system in Hertz.

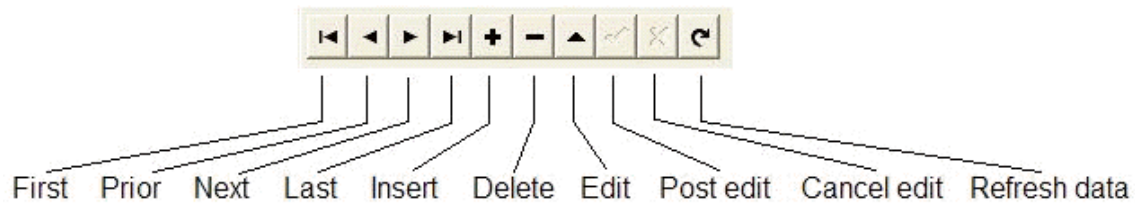
System of units: The system of units used to specify the values of the tower configuration. The options are English system or Metric (SI) system. The units specified here not necessarily have to match the units specified in the **Input Data** section.

### Database

The conductor type and tower configurations tables are read by default from the file **conductors.mdb**, which is a MS Access® database. This database can be read from another \*.mdb file by clicking on the **Select Conductors and Configurations Database** button.

Note: The conductors.mdb file can also be viewed and edited in MS Access®.

In order to edit a record in the database tables, the user can use the toolbar designed to do that. Following there is a figure showing this toolbar:



The **First**, **Prior**, **Next**, and **Last** buttons are used to move among records. The **Insert**, **Delete** and **Edit** buttons are used to insert, delete or edit the current record, respectively. While editing a record, the user can not change of tab until modifications are posted through the **Post edit** button or canceled with the **Cancel edit** button. The **Refresh data** button just refreshed the data of the entire table.

### Calculations

#### Distributed Parameters

##### Resistance

$$R_t = \frac{\left( R_{25} + \left( \frac{R_{25} - R_{50}}{25 - 50} \right) (t - 25) \right)}{N} \quad 0 \leq t \leq 50$$

$$R_t = \frac{\left( R_{50} + \left( \frac{R_{50} - R_{75}}{50 - 75} \right) (t - 50) \right)}{N} \quad 50 \leq t$$

Where:

$R_t$  AC resistance at temperature  $t$  per phase per 1 mile in Ohms

$t$  Assumed temperature in Celsius degrees

$R_{25}$  AC resistance of the conductor at 60 Hz and 25°C per 1 mile in Ohms

$R_{50}$  AC resistance of the conductor at 60 Hz and 50°C per 1 mile in Ohms

$R_{75}$  AC resistance of the conductor at 60 Hz and 75°C per 1 mile in Ohms

$N$  Number of conductors per phase

### Inductive Reactance

$$X_L = 4\pi f \times 10^{-7} \ln \frac{D_{eq}}{D_{SL}}$$

Where:

$X_L$  Inductive reactance in Ohms/meter

$f$  Frequency of the system in Hertz

$D_{eq}$  Geometric mean distance between phases in meters

$D_{SL}$  Geometric mean radius between conductors of one phase in meters

The **geometric mean distance** between phases is defined as:

$$D_{eq} = \sqrt[3]{d_{ab}d_{bc}d_{ca}}$$

Where:

$d_{ab}, d_{bc}, d_{ca}$  Distances between phases a-b, b-c, c-a, respectively in meters

The **geometric mean radius** between conductors of one phase is defined as:

$$D_{SL} = GMR \quad \text{For 1 stranded conductor}$$

$$D_{SL} = e^{-1/4} r \quad \text{For 1 solid cylindrical conductor}$$

$$D_{SL} = \left( \prod_{k=1}^N \prod_{m=1}^N d_{km} \right)^{\frac{1}{N^2}} \quad \text{For more than 1 conductor bundle}$$

Where:

$D_{SL}$  Geometric mean radius in meters

$r$  External radius of conductor in meters

$GMR$  Geometric mean radius given in tables for one stranded conductor

$d_{km}$  Distance between conductors  $k$  and  $m$  in meters.

Note: If  $k = m$ , then  $d_{km} = D_{SL}$  for one stranded or solid cylindrical conductor.

### Susceptance

$$B = 2\pi f \left( \frac{2\pi\epsilon}{\ln \left( \frac{D_{eq}}{D_{sc}} \right)} \right)$$

Where:

$B$  Susceptance in Siemens/meter

$f$  Frequency of the system in Hertz

$\epsilon$  Constant permittivity =  $8.85418 \times 10^{-12}$

$D_{eq}$  Geometric mean distance between phases, defined as above

$D_{sc}$  Geometric mean radius between conductors of one phase using external radius in meters

The geometric mean radius between conductors of one phase using external radius is defined as:

$$D_{sc} = r \quad \text{For 1 conductor}$$

$$D_{sc} = \left( \prod_{k=1}^N \prod_{m=1}^N d_{km} \right)^{\frac{1}{N^2}} \quad \text{For more than 1 conductor bundle}$$

Where:

$D_{sc}$  Geometric mean radius in meters

$r$  External radius of conductor in meters

$d_{km}$  Distance between conductors  $k$  and  $m$  in meters.

Note: If  $k = m$ , then  $d_{km} = r$ .

### Conductance

Assumed  $G = 0$

Where:

$G$  Conductance in Siemens/meter

## Lumped (Total) Parameters

**Resistance, Inductive Reactance, Conductance and Susceptance, using the equivalent  $\pi$  circuit (long line)**

$$Z' = R' + jX' = Z_c \sinh \gamma \ell$$

$$Y' = G' + jB' = \frac{2}{Z_c} \tanh \frac{\gamma \ell}{2}$$

Where:

- $Z'$  Total series impedance of line in Ohms
- $Y'$  Total series admittance of line in Siemens
- $R'$  Total series resistance of line in Ohms
- $X'$  Total series inductive reactance of line in Ohms
- $G'$  Total series conductance of line in Siemens
- $B'$  Total series susceptance of line in Siemens
- $Z_c$  Characteristic impedance in Ohms
- $\gamma$  Propagation constant in meters<sup>-1</sup>
- $\ell$  Line length in meters

The characteristic impedance and propagation constant are defined as:

$$Z_c = \sqrt{\frac{z}{y}}$$

$$\gamma = \sqrt{zy}$$

Where:

- $z$  Distributed series impedance in Ohms/meter
- $y$  Distributed series admittance in Siemens/meter

The **distributed series impedance** and **distributed series admittance** are defined as:

$$z = R + jX$$

$$y = G + jB$$

Where:

$R$  Distributed series resistance in Ohms/meter

$X$  Distributed series inductive reactance in Ohms/meter

$G$  Distributed series conductance in Siemens/meter

$B$  Distributed series susceptance in Siemens/meter

### Surge Impedance Loading

The surge impedance loading is defined as the power delivered by a lossless line to a load resistance equal to the surge (or characteristic) impedance  $Z_c$  and is given by:

$$P_{sll} = \frac{|V_N|^2}{Z_c}$$

Where:

$P_{sll}$  Total surge impedance loading in a three-phase line in VA

$V_N$  Line-line nominal voltage in Volts

### Base Values

#### Impedance Base

$$Z_B = \frac{(V_B^u)^2}{S_B^{3\phi}}$$

Where:

$Z_B$  Impedance base in Ohms

$S_B^{3\phi}$  Power base in VA

$V_B^u$  Line-line voltage base in Volts

#### Admittance Base

$$Y_B = \frac{1}{Z_B}$$

Where:

$Y_B$  Admittance base in Siemens

$Z_B$  Impedance base in Ohms

### Per Unit (PU) Parameters



### Resistance, Inductive Reactance, Conductance, Susceptance

$$R_{PU} = \frac{R'}{Z_B}$$

$$X_{PU} = \frac{X'}{Z_B}$$

$$G_{PU} = \frac{G'}{Y_B}$$

$$B_{PU} = \frac{B'}{Y_B}$$

Where:

$R_{PU}$  Per unit resistance

$R'$  Total series resistance in Ohms

$X_{PU}$  Per unit Inductive reactance

$X'$  Total series inductive reactance in Ohms

$G_{PU}$  Per unit conductance

$G'$  Total series conductance in Siemens

$B_{PU}$  Per unit susceptance

$B'$  Total series susceptance in Siemens

$Z_B$  Impedance base in Ohms

$Y_B$  Admittance base in Siemens

### MVA To Ampere and Ampere To MVA Limits Conversion

MVA to Ampere Limit Conversion

$$Lim_{Amp} = \frac{Lim_{MVA} \times 10^6}{\sqrt{3} |V_N|}$$

Where:

$Lim_{Amp}$  Limit in Amperes

$Lim_{MVA}$  Limit in MVAs

$V_N$  Nominal voltage in Volts

### Ampere to MVA Limit Conversion

$$Lim_{MVA} = Lim_{Amp} \frac{\sqrt{3} |V_N|}{10^6}$$

Where:

$Lim_{Amp}$  Limit in Amperes

$Lim_{MVA}$  Limit in MVAs

$V_N$  Nominal voltage in Volts

---

## Transmission Line/Transformer Options: Transformer Control

---

Transformers are used to transfer power between different voltage levels or to regulate real or reactive flow through a particular transmission corridor. Most transformers come equipped with taps on the windings to adjust either the voltage transformation or the reactive flow through the transformer. Such transformers are called either load-tap-changing (LTC) transformers or tap-changing-under-load (TCUL) transformers.

Another type of transformer is known as a phase-shifting transformer (phase shifter or phase angle regulator). Phase-shifting transformers, which are less common than LTC transformers, vary the angle of the phase shift across the transformer in order to control the MW power flow through the transformer.

### Off-nominal Turns Ratio and Phase Shift Degrees

The [Line/Transformer Dialog](#) displays several transmission line and transformer properties. For transformers, this dialog box also shows information about the LTC or phase shifter controls. The **Off-nominal Turns Ratio** field indicates the voltage transformation, while the **Phase Shift Degrees** field show the phase shift angle. If the transformer is not on automatic control, these values can be changed manually. The off-nominal tap ratio determines the additional transformation relative to the nominal transformation. This value normally ranges from 0.9 to 1.1 (1.0 corresponds to no additional transformation). For phase-shifting transformers the phase shift value normally ranges from about -40° to 40°. The phase angle field can be non-zero for LTC and fixed transformers, most notably +/- 30° if the transformer configuration is a delta-wye or wye-delta configuration. The transformer configuration is very important when performing a [fault analysis](#) study.

When in Edit Mode, the dialog also reveals the type of transformer. Valid types are 1) No Automatic Control (in which the taps are assumed fixed), 2) voltage regulation (AVR), 3) Reactive Power Control, and 4) Phase Shift Control. The type of transformer CANNOT be modified in the Run Mode.

Simulator provides you with a great deal of flexibility in being able to specify which transformers will actually be used for automatic control in the Power Flow Solution. For a transformer to be used for voltage or flow control, three criteria must be met.

- The transformer's **Automatic Control Enabled** field must be checked on its [Line/Transformer dialog](#). This field can also be modified on the [Transformer Records display](#).
- The transformer's area must have automatic transformer control enabled. This is specified on the *Options Tab* of the [Area Records](#) display.
- Transformer control must not be disabled for the entire case. This is specified on the *Power Flow Solution Tab* of the [Simulator Options Dialog](#).

The area and case enforcement of transformer control are also accessible from the Run Mode [Line/Transformer Dialog](#).

### Automatic Control

The **Automatic Control** fields are only visible in Edit Mode when the [Change Automatic Control Options](#) button is pressed.

### No Automatic Control

On this control setting the transformer will operate at the given off-nominal turns ratio and phase shift, and will remain fixed at those values during the entire solution process unless manually changed by the user.

### AVR (Automatic Voltage Regulation)

When on automatic voltage control, the transformer taps automatically change to keep the voltage at the regulated bus (usually one of the terminal buses of the transformer) within a voltage range between the minimum voltage and maximum voltage values (given in per unit). These values can be seen by clicking on the [Automatic Control Options](#) button. Note that automatic control is possible only if a regulated bus has been specified.

The tap position for an LTC transformer is indicated on the oneline by the number of tap step positions from the nominal position (i.e., the position when the off-nominal tap ratio is equal to 1.0). When the off-nominal ratio is greater than 1.0, the transformer's tap is said to be in the "raise" position, and an 'R' appears after the number. Likewise, when the off-nominal ratio is less than 1.0, the transformer's tap is said to be in the "lower" position, and an 'L' appears after the number. For example, with a step size of 0.00625 and an off-nominal ratio of 1.05, the tap would be in position 8R. The tap position can be changed manually only when the transformer has been set off automatic voltage control.

Simulator will also detect instances when controlling transformers are in parallel, and will employ checks during the solution routine to prevent the controllers from fighting each other and potentially going to opposite tap solutions, which could result in unwanted loop flow through the transformer objects. This option is enabled by default, but can be turned off in the [Power Flow Solution General Options](#).

### Transformer Reactive Power Control

When on automatic reactive power control, the transformer taps automatically change to keep the reactive power flow through the transformer (measured at the *from bus*) within a user-specified range. The reactive power control parameters can be seen by clicking the [Automatic Control Options](#) button.

### Phase Shift Control

When a transformer is on phase shift control, the transformer phase shift angle automatically changes to keep the MW flow through the transformer (measured at the *from bus*) between the minimum and maximum flow values (given in MW, with flow into the transformer assumed positive). The limits on the phase shifting angles are specified in the minimum and maximum phase fields (in degrees). These values can be seen by clicking on the [Automatic Control Options](#) button. The phase shift angle changes in discrete steps, with the step size specified in the Step Size field (in degrees). The **MW Per Phase Angle Step Size** provides an estimate of the change in the controlled MW flow value if the phase angle is increased by the step size value.

### Specify Transformer Bases and Impedances

Shows the Transformer Bases and Impedances Dialog. This dialog allows the user to specify the transformer parameters in per unit on the transformer base (taken as its rating). Click **OK** to convert all the transformer parameters values to the system base specified in the [General Power Flow Solution Options](#).

---

## Transformer AVR Dialog

---

The Transformer AVR Dialog is used to view the control parameters associated with load-tap-changing (LTC) transformers when they are used to control bus voltage magnitudes. To view this display, click the *Automatic Control Options* button on the [Line/Transformer Information Dialog](#).

This dialog has the following fields:

### Automatic Control Type

This section is only visible on the dialog in edit mode. The control type of the transformer can be changed between no control, voltage regulation, reactive power control, or phase shifter control.

### Regulated Bus Number

The number of the bus whose voltage is regulated by the control.

### Present Regulated Bus Voltage

The present voltage of the regulated bus.

### Voltage Error

If the regulated bus' voltage falls outside the regulating range of the transformer (as defined by the *Minimum Voltage* and *Maximum Voltage* fields), the *Voltage Error* field indicates by how much the voltage deviates from the control range.

### Regulation Minimum Voltage

The minimum acceptable voltage at the regulated bus.

### Regulation Maximum Voltage

The maximum acceptable voltage at the regulated bus.

### Regulation Target Type

As long as the regulated voltage is inside the regulation minimum and maximum, then the transformer will not change its tap ratio. When the regulated voltage moves outside of this regulation range, then Simulator will calculate a new tap ratio in an attempt to bring the regulated voltage back inside of its range. The **Regulation Target Type** determines what value is used as a target when calculating this change in tap ratio. **Middle** is the default and means that the target is the average of the regulation minimum and maximum regardless of whether the voltage is high or low. **Max/Min** means that the regulation maximum is used as the target when the regulated voltage is above the maximum, and regulation minimum is used as the target value when the regulated voltage is below the minimum.

### Present Tap Ratio

The tap ratio of the transformer for the current system state.

### Minimum Tap Ratio, Maximum Tap Ratio

Minimum and maximum allowable off-nominal tap ratios for the LTC transformer. Typical values are 0.9 and 1.1.

### Tap Step Size

Transformer off-nominal turns ratio increment. The off-nominal turns ratio is either incremented or decremented from 1.0 in integer multiples of this value. Default value is 0.00625.

### Voltage to Tap Sensitivity

Shows the sensitivity of the voltage magnitude at the regulated bus to a change in the transformer's tap ratio. You can use this field to assess whether or not the transformer can effectively control the regulated bus voltage. In an ideal case, such as when the LTC transformer is being used to control the voltage at a radial load bus, the sensitivity is close to 1.0 (or -1.0 depending upon whether the tapped side of the transformer is on the load side or opposite side of the transformer). However, sometimes the transformer is very ineffective in controlling the voltage. This is indicated by the absolute value of the sensitivity approaching 0. A common example is a generator step-up transformer trying to control its high-side voltage when the generator is off-line. Simulator automatically disables transformer control if the transformer sensitivity is below the value specified on *Power Flow Solution Tab* of the [Simulator Options dialog](#).

### Impedance Correction Table

This field specifies the number of the transformer's corresponding transformer impedance correction table. Transformer impedance correction tables are used to specify how the impedance of the transformer should change with the off-nominal turns ratio. If this number is 0, then no impedance correction table is associated with the transformer, and the impedance of the transformer will thus remain fixed as the tap ratio changes. Valid impedance correction table numbers range from 1 to 63. To assign an existing impedance correction table to the transformer, enter the existing table's number. To view the existing impedance correction tables, click the *Insert/View Impedance Correction Table* button, which brings up the [Transformer Impedance Correction Dialog](#). To define a brand new impedance correction table for the transformer, enter an unused table number and then click *Insert/View Impedance Correction Table* to prescribe the correction table. Note that the association between a transformer and an impedance correction table is not finalized until you select either *OK* or *Save* on the [Line/Transformer Dialog](#).

### View Transformer Correction Table or Insert Transformer Correction Table

Click on this button either to view or to insert transformer correction tables. Clicking on this button displays the [Transformer Impedance Correction Dialog](#). Note that the table must prescribe at least two points in order to be defined.

---

**Transmission Line/Transformer Options: Series Capacitor**

---

The Series Capacitor tab of the Line/Transformer Information dialog displays information relating to a series capacitor, including its status.

**Status**

The capacitor itself has two status positions, **Bypassed** and **In Service**. When the series capacitor is in service, the branch is modeled as a reactive branch, using the line parameters from the Parameters / Display page. If the capacitor is bypassed, a low impedance branch is introduced to bypass the capacitor. Note that this is not the same as the branch status of **Open** or **Closed**. The branch status is the indicator of whether or not the entire circuit is operating (closed), regardless of **Bypassed** or **In Service** status on the capacitor itself.

**Is Series Capacitor**

If this box is checked, the branch can be treated as a series capacitor, with the series Status of Bypassed or In-Service available.

---

## Multi-Section Line Information

---

Multi-section line records are used to group a number of series-connected transmission lines together so that their status is controlled as a single entity. They are usually used to model very long transmission lines that require multiple individual transmission line records if they are to be modeled accurately. In terms of line status, Simulator then treats each multi-section line as a single device. In other words, changing the status of one line in the record changes the status of the other lines in the record, as well.

Multi-section line records are **not** directly represented on the oneline diagrams. Rather, to view this dialog, right-click the record of interest on the [Multi-Section Line Records](#) display and select **Show Dialog** from the resulting local menu. To define a new multi-section line, switch to Edit Mode and, from the Multi-Section Line Records display, right-click and select **Insert** from the resulting local menu.

**Note:** If any of the transmission lines comprising a multi-section line record are deleted, the entire multi-section line record is deleted as well.

The Multi-Section Lines Dialog contains the following fields:

### From Bus Number and Name

Number and name of the from bus for the record. If you are defining a new multi-section line, this must be the first data item you specify.

### To Bus Number and Name

Number and name of the "to" bus for the record. You cannot specify this value directly. Instead, use the table at the bottom of the dialog to define the intermediate buses and *to* terminal.

### Circuit

Two character identifier used to distinguish between multiple records joining the same from/to buses. The first character of the circuit identifier must be an "&."

You can use the spin button immediately to the right of the circuit field to view other multi-section line records. However if you have changed the record, **you must select Save** before moving to another record. Otherwise, your changes will be lost.

### From End Metered

If checked, the *from end* of the record is the metered end; otherwise the *to end* is metered.

### Multi-Section Line Name

The name for the multi-section line.

### Table

The table lists the dummy (or intermediate) buses and the *to bus* that comprise the record. The first column should contain the first dummy bus number and the circuit ID of the transmission line joining the *from bus* with the first dummy bus. The next column should contain the second dummy bus number and the circuit id of the transmission line joining this bus to the first dummy bus. Continue until the last column contains the *to* bus and the circuit ID of the transmission line joining the last dummy bus with the "to" bus.

Each transmission line comprising the multi-section line must already exist. If only one transmission line joins any two buses, you may omit the circuit identifier.

### OK

Saves any modifications and closes the dialog.



**Save**

Saves any modifications but does not close the dialog.

**Cancel**

Closes the dialog without saving modifications to the current record.

**Delete**

Deletes the current multi-section line record.

**Note:** If any of the transmission lines comprising a multi-section line record are deleted, the entire multi-section line record is deleted as well.

---

## Line Field Options

---

Line field objects are used to show different values associated with transmission lines, transformers and DC lines. This dialog is used to view and modify the parameters associated with these fields.

### Find...

If you do not know the exact line identifiers you are looking for, you can click this button to open the [advanced search engine](#).

### Near Bus Number

Bus associated with the near end of the object. All fields display values calculated at the *near bus* end. When inserting fields graphically, this field is automatically set to the closest bus on the oneline.

### Far Bus Number

Bus associated with the *far end* of the object.

### Circuit

Two-character identifier used to distinguish between multiple lines or transformers joining the same two buses. Default is '1'.

### Total Digits in Fields

Total number of digits to show in the field.

### Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

### Delta Per Mouse Click

Only used with the MVA Limit field type. Specifying a nonzero value for this field equips the MVA Limit Field with an integrated spin button that can be clicked to increment or decrement the MVA Limit by the amount of the *Delta Per Mouse Click* value.

### Field Value

The current value of the field being displayed.

### Field Prefix

A prefix that can be specified and displayed with the selected value.

### Rotation Angle in Degrees

The angle at which the text will appear on the diagram.

### Anchored

If checked, the line analog is [anchored](#) to its associated line.

### Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

### Type of Field

Used to determine the type of line field to show. The following choices are available:

**AC Line MW Flow** MW flow into the line or transformer at the near bus

**AC Line Mvar Flow** Mvar flow into the line or transformer at the near bus

**AC Line MVA Flow** Magnitude of MVA flow into the line or transformer at the near bus

**AC Line Amp Flow** Magnitude of Amp flow (in amps) into the line at the near bus

**AC Line MW Losses** Real power losses on the line or transformer in MW

**AC Line Mvar Losses** Reactive power losses on the line or transformer in Mvar

**DC Line MW Flow** MW flow into the DC line at the near bus

**DC Line Mvar Flow** Mvar flow into the DC line at the near bus

**MVA Limit** MVA limit for the line or transformer

**DC Line Set point** Set point value for the DC line; see [DC Transmission Line dialog](#)

**Select a Field** Choose from over 80 different fields

Select **OK** to save changes and close the dialog or **Cancel** to close the dialog without saving your changes.

---

## Series Capacitor Field Options Dialog

---

Series capacitor field objects are used to show field values specific to series capacitors. Use [Line Fields](#) to show fields generic to transformers, series capacitors and transmission lines, such as the flow of power through the device. The series capacitor fields dialog is used to view and modify the parameters associated with series capacitor specific fields.

### Near Bus Number

Bus associated with the *near end* of the series capacitor.

### Far Bus Number

Bus associated with the *far end* of the series capacitor.

### Circuit

Two-character identifier used to distinguish between branches joining the same two buses. Default is '1'.

### Anchored

When checked, the text field will move with the series capacitor if the series capacitor is moved on the oneline diagram.

### Total Digits in Field

Total number of digits to show in the field.

### Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

### Rotation Angle in Degrees

The angle at which the text will be placed.

### Field Value

The value of the currently selected field.

### Field Prefix

A prefix that can be specified and displayed with the selected value.

### Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

### Type of Field

Designates the type of transformer field to show. The following choices are available:

**Status** Capacitor status

**Series Capacitance** Capacitance of series capacitor

Select **OK** to save changes and close the dialog or **Cancel** to close dialog without saving your changes.

---

## Transformer Field Options Dialog

---

Transformer field objects are used to show field values specific to transformers. Use [Line Fields](#) to show fields generic to transformers and transmission lines, such as the flow of power through the device. The transformer fields dialog is used to view and modify the parameters associated with transformer-specific fields.

### Near Bus Number

Bus associated with the *near end* of the transformer.

### Far Bus Number

Bus associated with the *far end* of the transformer.

### Circuit

Two-character identifier used to distinguish between transformers joining the same two buses. Default is '1'.

### Find...

If you do not know the exact transformer you are looking for, you can click this button to open the [advanced search engine](#).

### Total Digits in Field

Total number of digits to show in the field.

### Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

### Delta per Mouse Click

This value is used only with the **Off-nominal Tap Ratio** and **Phase Shift Angle** field types. When there is a nonzero entry in this field, and the field type is valid, a spin button is shown to the right of the zone field. When the up spin button is clicked, the field value is increased by this number; when the down button is clicked, the field value is decreased by this amount.

### Field Value

Shows the current output for the transformer field. Whenever you change the **Type of Field** selection, this field is updated.

### Field Prefix

A prefix that can be specified and displayed with the selected value.

### Rotation Angle in Degrees

The angle at which the text is to appear on the oneline diagram.

### Anchored

When checked, the text field will move with the transformer if the transformer is moved on the oneline diagram.

### Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

### Type of Field

Designates the type of transformer field to show. The following choices are available:

**Off-nominal Tap Ratio** Actual tap ratio

**Phase Shift Angle** Actual phase shift in degrees

**Off-nominal Tap Position** Tap position in steps, usually ranging from L16 to R16

**Automatic Control Status** The status of the control for the transformer

Select **OK** to save changes and close the dialog or **Cancel** to close dialog without saving your changes.

---

## DC Transmission Line Options

---

This dialog is used to view and modify the parameters associated with each two-terminal DC transmission line in the system. It can also be used to insert new two-terminal DC transmission lines or to delete existing ones.

This dialog has six separate pages: [Line Parameters](#), [Rectifier Parameters](#), [Inverter Parameters](#), [Actual Flows](#), [OPF](#) and [Custom](#). The Actual Flows page is not used in Edit Mode. The separate pages can be accessed using the tabs shown at the top of the dialog. These pages can be used to view/change the modeling parameters associated with the DC lines.

---

## DC Line Options: Line Parameters

---

This page is used to enter parameters associated with the DC line itself.

### Rectifier and Inverter Bus Numbers and Names

These fields indicate the numbers and names of the rectifier and inverter ends of the line. When graphically inserting a DC line, these fields are automatically determined based upon the starting and ending buses used in drawing the line. When investigating existing DC line records, you may use the *Find By Numbers* button to identify a DC line between a specific rectifier - inverter pair. You may also use the spin control to cycle through the list of DC line records modeled in the case.

### Circuit ID

This field is not currently used, since only one DC transmission line can exist between each inverter-rectifier pair. It is hard-coded to be 1.

### Find By Numbers

To find a DC line by its bus numbers, enter the *Rectifier* and *Inverter* bus numbers and the circuit identifier. Then click this button. Use the spin button to cycle through the list of DC lines in the system.

### Area Name

Names of the areas in which the rectifier and inverter buses are located.

### Labels

Clicking this button will open the [Subscribed Aliases dialog](#) displaying the list of defined labels for the DC line. New labels can also be added for the DC Line from the dialog as well.

### Link to New DC Line

Use the *Link to New DC Line* button to create a new DC line corresponding to the entries you have made in the dialog. This button performs the same function as pressing *Save*. Note that adding a new DC line to the case in this way does not add a DC line to the online display; the new DC line is present only in the model. You may then add the newly modeled DC line to the online diagram in the usual way (such as select **Network > DC Transmission Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab).

## Line Parameters

### Status

Operating status of the DC transmission line. If the status is Open, then no power can flow on the DC line regardless of the control mode setting. If the status is closed, then the control mode will dictate how power flows on the line.

### Control Mode

The initial control mode for the line. Specify *Blocked* to disable the DC line, *Power* to maintain specified MW power flow through the line, or *Current* to maintain specified current flow through the line.

### Setpoint

If the line operates in the Power Control Mode, *Setpoint* should indicate the desired power flow in MW. To specify power flow at the rectifier end, enter a positive value. Enter a



negative value to specify power flow at the inverter end. If the DC line operates in Current Control Mode, enter the desired line flow in amps.

**Resistance**

Resistance of the DC Line.

**Sched. Voltage**

Scheduled DC line voltage in kV. The value of Rcomp is used to determine whether this value specifies the inverter end or the rectifier end.

**Switch Voltage**

When the line operates in the Power Control Mode, this is the inverter voltage level in kV at which the line switches from constant power to constant current control.

**RComp**

Compounding resistance. The compounding resistance dictates whether the inverter voltage is scheduled ( $R_{comp} = 0$ ), or the rectifier voltage is scheduled ( $R_{comp} = \text{DC line resistance}$ ). Simulator does not currently support scheduling the voltage at a point along the DC line ( $0 < R_{com} < \text{DC line resistance}$ ), but instead will treat any non-zero value of Rcomp as scheduling the voltage at the rectifier.

**Setpoint Specified At**

Indicates which end of the DC transmission line the Setpoint value is designated. This will be the terminal where the setpoint value is maintained. The opposite terminal flow value will be a calculated quantity.

**Metered End of Line**

Indicates which end of the DC line is assumed metered for Area interchange calculations.

---

**DC Line Options: Rectifier Parameters**

---

This page is used to enter parameters associated with the rectifier end of the line.

**# of Bridges**

Number of valve bridges in series.

**Base Voltage**

Base AC voltage in kV on primary side of transformer.

**XF Ratio**

Transformer ratio.

**XF Tap**

Transformer tap setting.

**XF Min/Max Tap, XF Tap Step**

Transformer minimum and maximum tap settings, and the tap's step size.

**Commuting XF Resistance and Reactance**

Commuting resistance and reactance for the transformer, in ohms.

**Minimum, Maximum, and Actual Firing Angle**

Minimum, maximum, and actual values of the firing angle for the rectifier.

---

**DC Line Options: Inverter Parameters**

---

This page is used to enter parameters associated with the inverter end of the line. Entries are identical to the Rectifier Page, except here they are associated with the inverter.

**# of Bridges**

Number of valve bridges in series.

**Base Voltage**

Base AC voltage in kV on primary side of transformer.

**XF Ratio**

Transformer ratio.

**XF Tap**

Transformer tap setting.

**XF Min/Max Tap, XF Tap Step**

Transformer minimum and maximum tap settings, and the tap's step size.

**Commuting XF Resistance and Reactance**

Commuting resistance and reactance for the transformer, in ohms.

**Minimum, Maximum, and Actual Firing Angle**

Minimum, maximum, and actual values of the firing angle for the rectifier.

---

**DC Line Options: Actual Flow**

---

Shows the actual real and reactive power flows into the line at the rectifier and the inverter, as well as the voltage at both ends (in kV) and the line current in amps. The contents of this page are valid only when Simulator is in Run Mode.

---

**DC Line Options: OPF**

---

This page is used to enter parameters associated with the OPF.

**Rectifier/Inverter Bus MW Marginal Costs**

Displays the marginal costs of the branches terminal buses, following the solution of the OPF.

**OPF Control****Minimum and Maximum Setpoint Value**

The minimum and maximum setpoint allowed for the DC Line. When on OPF Control the DC Line will be dispatched between these values.

**Current Setpoint Value**

The current Setpoint as found on the Line Parameters Tab.

**Transmission Charge**

This is the cost incurred for moving power across the DC transmission line. The cost is zero at a zero MW flow and increases linearly using this charge.

**OPF Control Enabled for this DC line**

This check box must be checked if the DC limit is going to be enforced when running an OPF solution. If this box is not checked, the OPF routine will allow the DC line to violate its limits.

**Include Impact of DC Line Marginal Losses**

When dispatching the DC Line this determines whether the marginal losses from the DC line are taken into account.

---

## Multi-Terminal DC Record Information

---

This dialog is used to view and modify the parameters associated with multi-terminal DC records. It is also used when inserting new multi-terminal DC records.

The Multi-Terminal DC Record dialog can be used to inspect and modify the model of a multi-terminal DC network record. To view the Multi-Terminal DC Record, simply right-click on the record of interest in the [Multi-Terminal DC Record Display](#) and select *Show Dialog* from the resulting popup menu. The dialog has the following fields:

### Record Number

Unique number between 1 and 999 which identifies the current multi-terminal DC record.

### Number of Devices

Lists the number of DC buses, converters, and DC Lines that form the multi-terminal DC network.

### Control

The control method used when solving the multi-terminal DC network.

### Controlling Converter

The AC converter bus number where the DC voltage is being controlled.

### MTDC Network Status

Status of the entire Multi-terminal DC network. If this field is set to Closed, the entire DC subnetwork of the MTDC model is considered disconnected in the load flow case.

### DC Buses Tab

This page of the dialog displays the DC bus records for the multi-terminal DC network. The display on this page exhibits the same features as other [case information displays](#). To view the specific information for a DC bus, right-click on the record of interest and choose *Show Dialog* from the popup menu.

### DC Converters Tab

This page of the dialog displays the DC converter records for the multi-terminal DC network. The display on this page exhibits the same features as other [case information displays](#). To view the specific information for a DC converter, right-click on the record of interest and choose *Show Dialog* from the popup menu.

### DC Lines Tab

This page of the dialog displays the DC line records for the multi-terminal DC network. The display on this page exhibits the same features as other [case information displays](#). To view the specific information for a DC line, right-click on the record of interest and choose *Show Dialog* from the popup menu.

### Custom Tab

Enter any text notes you wish in the Memo page. When the case is saved as a Simulator PWB file, the memo text will also be saved. Custom strings and values can also be entered and saved with the case file as well.

---

**Multi-Terminal DC Bus Information**

---

This dialog is used to view and modify the parameters specific to multi-terminal DC network buses. The dialog is also used to enter values for new multi-terminal DC buses when inserting a [Multi-Terminal DC Record](#).

To view the Multi-Terminal DC Bus Record dialog, right-click on a bus record in the DC Buses tab of the [Multi-Terminal DC Record dialog](#) and select *Show Dialog* from the resulting popup menu. The dialog has the following fields:

**DC Bus Number, DC Bus Name**

The number and name identifiers for the selected DC bus.

**AC Bus Number**

The AC bus number connected to the DC bus through an AC/DC converter. If the selected DC bus is a bus that is internal to the DC multi-terminal network (not directly connected to an AC bus,) this field will be 0.

**Area Number, Area Name**

The number and name identifiers of the control area the DC bus is contained in.

**Zone Number, Zone Name**

The number and name identifiers of the zone the DC bus is contained in.

**Ground Resistance**

Resistance to ground of the DC bus, entered in Ohms. This field is currently only for storage of values supported by other load flow formats, and is currently not used by Simulator.

---

## Multi-Terminal DC Converter Information

---

This dialog is used to view and modify the parameters specific to multi-terminal DC network buses. The dialog is also used to enter values for new multi-terminal DC buses when inserting a [Multi-Terminal DC Record](#).

To view the Multi-Terminal DC Bus Record dialog, right-click on a bus record in the DC Buses tab of the [Multi-Terminal DC Record dialog](#) and select *Show Dialog* from the resulting popup menu. The dialog has the following fields:

### Converter Parameters

#### Number of Bridges

Number of bridges in series for the selected converter.

#### Converter Type

**R** for rectifier or **I** for inverter.

#### Commutating Impedance

Commutating impedance per bridge, in Ohms.

#### Firing Angle Limits

The maximum and minimum firing angle limits, in degrees.

### Transformer Parameters

#### AC Base

The primary AC base voltage, in kV.

#### DC Base

The DC base voltage, in kV.

#### Transformer Ratio

Actual transformer ratio.

#### Tap Settings

Displays the actual tap setting, the tap step, and maximum and minimum tap values for the converter transformer.

### Control Parameters

#### Setpoint

The setpoint control value at the converter. For the voltage-controlling converter, this field is set to 0. For the remaining converters, this field displays MW when in Power mode, or Amps when in Current mode.

#### Margin

Rectifier margin, entered in per-unit of the DC power or current. This field is currently only for support of other load flow formats, and is not used by Simulator.

#### DC Participation Factor

Converter participation factor. This field is currently only for support of other load flow formats, and is not used by Simulator.



**Voltage**

The DC Voltage magnitude at the DC side of the converter.

**Solved Parameters**

**Firing Angle**

The firing angle of the converter, as determined during the load flow solution.

**DC Current**

The calculated DC current at the converter DC terminal.

**MW, MVAR**

The real and reactive power delivered to (or absorbed from) the AC system by the converter.

---

## Multi-Terminal DC Line Information

---

This dialog is used to view and modify the parameters specific to multi-terminal DC network buses. The dialog is also used to enter values for new multi-terminal DC buses when inserting a [Multi-Terminal DC Record](#).

To view the Multi-Terminal DC Bus Record dialog, right-click on a bus record in the DC Buses tab of the [Multi-Terminal DC Record dialog](#) and select *Show Dialog* from the resulting popup menu. The dialog has the following fields:

### From and To DC Bus Number

The DC bus numbers of the From and To buses in the multi-terminal DC network. These fields must contain valid DC bus numbers of the selected multi-terminal DC record. AC bus numbers from the load flow case are not acceptable bus numbers for a multi-terminal DC line.

### DC Circuit

The circuit identifier for the DC line.

### DC Resistance and DC Inductance

The resistance and inductance of the DC line. Resistance is in Ohms, and is used for solving the load flow of the DC network. The inductance is in milliHenries, and is not used for solving the load flow. The inductance field is currently only for support of other load flow formats, and is not used by Simulator.

---

## Transmission Line/Transformer Options: Transformer Control

---

Transformers are used to transfer power between different voltage levels or to regulate real or reactive flow through a particular transmission corridor. Most transformers come equipped with taps on the windings to adjust either the voltage transformation or the reactive flow through the transformer. Such transformers are called either load-tap-changing (LTC) transformers or tap-changing-under-load (TCUL) transformers.

Another type of transformer is known as a phase-shifting transformer (phase shifter or phase angle regulator). Phase-shifting transformers, which are less common than LTC transformers, vary the angle of the phase shift across the transformer in order to control the MW power flow through the transformer.

### Off-nominal Turns Ratio and Phase Shift Degrees

The [Line/Transformer Dialog](#) displays several transmission line and transformer properties. For transformers, this dialog box also shows information about the LTC or phase shifter controls. The **Off-nominal Turns Ratio** field indicates the voltage transformation, while the **Phase Shift Degrees** field show the phase shift angle. If the transformer is not on automatic control, these values can be changed manually. The off-nominal tap ratio determines the additional transformation relative to the nominal transformation. This value normally ranges from 0.9 to 1.1 (1.0 corresponds to no additional transformation). For phase-shifting transformers the phase shift value normally ranges from about -40° to 40°. The phase angle field can be non-zero for LTC and fixed transformers, most notably +/- 30° if the transformer configuration is a delta-wye or wye-delta configuration. The transformer configuration is very important when performing a [fault analysis](#) study.

When in Edit Mode, the dialog also reveals the type of transformer. Valid types are 1) No Automatic Control (in which the taps are assumed fixed), 2) voltage regulation (AVR), 3) Reactive Power Control, and 4) Phase Shift Control. The type of transformer CANNOT be modified in the Run Mode.

Simulator provides you with a great deal of flexibility in being able to specify which transformers will actually be used for automatic control in the Power Flow Solution. For a transformer to be used for voltage or flow control, three criteria must be met.

- The transformer's **Automatic Control Enabled** field must be checked on its [Line/Transformer dialog](#). This field can also be modified on the [Transformer Records display](#).
- The transformer's area must have automatic transformer control enabled. This is specified on the *Options Tab* of the [Area Records](#) display.
- Transformer control must not be disabled for the entire case. This is specified on the *Power Flow Solution Tab* of the [Simulator Options Dialog](#).

The area and case enforcement of transformer control are also accessible from the Run Mode [Line/Transformer Dialog](#).

### Automatic Control

The **Automatic Control** fields are only visible in Edit Mode when the [Change Automatic Control Options](#) button is pressed.

### No Automatic Control

On this control setting the transformer will operate at the given off-nominal turns ratio and phase shift, and will remain fixed at those values during the entire solution process unless manually changed by the user.

### AVR (Automatic Voltage Regulation)

When on automatic voltage control, the transformer taps automatically change to keep the voltage at the regulated bus (usually one of the terminal buses of the transformer) within a voltage range between the minimum voltage and maximum voltage values (given in per unit). These values can be seen by clicking on the [Automatic Control Options](#) button. Note that automatic control is possible only if a regulated bus has been specified.

The tap position for an LTC transformer is indicated on the oneline by the number of tap step positions from the nominal position (i.e., the position when the off-nominal tap ratio is equal to 1.0). When the off-nominal ratio is greater than 1.0, the transformer's tap is said to be in the "raise" position, and an 'R' appears after the number. Likewise, when the off-nominal ratio is less than 1.0, the transformer's tap is said to be in the "lower" position, and an 'L' appears after the number. For example, with a step size of 0.00625 and an off-nominal ratio of 1.05, the tap would be in position 8R. The tap position can be changed manually only when the transformer has been set off automatic voltage control.

Simulator will also detect instances when controlling transformers are in parallel, and will employ checks during the solution routine to prevent the controllers from fighting each other and potentially going to opposite tap solutions, which could result in unwanted loop flow through the transformer objects. This option is enabled by default, but can be turned off in the [Power Flow Solution General Options](#).

### Transformer Reactive Power Control

When on automatic reactive power control, the transformer taps automatically change to keep the reactive power flow through the transformer (measured at the *from bus*) within a user-specified range. The reactive power control parameters can be seen by clicking the [Automatic Control Options](#) button.

### Phase Shift Control

When a transformer is on phase shift control, the transformer phase shift angle automatically changes to keep the MW flow through the transformer (measured at the *from bus*) between the minimum and maximum flow values (given in MW, with flow into the transformer assumed positive). The limits on the phase shifting angles are specified in the minimum and maximum phase fields (in degrees). These values can be seen by clicking on the [Automatic Control Options](#) button. The phase shift angle changes in discrete steps, with the step size specified in the Step Size field (in degrees). The **MW Per Phase Angle Step Size** provides an estimate of the change in the controlled MW flow value if the phase angle is increased by the step size value.

### Specify Transformer Bases and Impedances

Shows the Transformer Bases and Impedances Dialog. This dialog allows the user to specify the transformer parameters in per unit on the transformer base (taken as its rating). Click **OK** to convert all the transformer parameters values to the system base specified in the [General Power Flow Solution Options](#).

---

**Transformers Bases and Impedances Dialog**

---

Typically the impedances and tap values of transformers is already assumed to have been converted to unity tap base and bus nominal voltage base. However, some load flow formats provide the taps and impedances on specific transformer bases, which are different than the bus voltage and unity tap base assumptions. In these cases, Simulator will convert parameters from the transformer bases to the unity tap and bus nominal voltage base. Display of the impedances and tap values normally displayed in the [Line/Transformer Options](#) dialog are displayed on the Simulator assumed bases. However, if you wish to view the original transformer values on the transformer supplied bases, this dialog will display the original values. You can modify the original values stored here in this dialog. Note that when you do so, the converted values that Simulator stores on the system bases will also be automatically updated to reflect the change that has been made to the original values on the transformer bases.

---

## Transformer Impedance Correction Table Display

---

The Transformer Impedance Correction Display shows information about all the transformer impedance correction tables in the case. The Transformer Impedance Correction Display is used to model the change in the impedance of the transformer as the off-nominal turns ratio or phase shift angle is varied.

The Correction Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated correction tables. When in Edit Mode, you can define new tables using the *Insert* option, or delete existing tables using *Delete*. You can also sort the transformer impedance correction information by clicking on the heading of the field by which you want to sort.

To show this display select **Network > Impedance Correction Tables** from the [Model Explorer](#).

The display contains the following fields by default:

### Table #, Table Name

Shows the table number and name for the record. Table number must be between 1 and 64.

### Transformer Impedance Scaling Factors

The rest of the default columns show the actual fields in the table. The first field of each pair shows the off-nominal turns ratio or phase shift angle, while the second field shows the associated scaling factor for the transformer's impedance.

---

## Transformer AVR Dialog

---

The Transformer AVR Dialog is used to view the control parameters associated with load-tap-changing (LTC) transformers when they are used to control bus voltage magnitudes. To view this display, click the *Automatic Control Options* button on the [Line/Transformer Information Dialog](#).

This dialog has the following fields:

### Automatic Control Type

This section is only visible on the dialog in edit mode. The control type of the transformer can be changed between no control, voltage regulation, reactive power control, or phase shifter control.

### Regulated Bus Number

The number of the bus whose voltage is regulated by the control.

### Present Regulated Bus Voltage

The present voltage of the regulated bus.

### Voltage Error

If the regulated bus' voltage falls outside the regulating range of the transformer (as defined by the *Minimum Voltage* and *Maximum Voltage* fields), the *Voltage Error* field indicates by how much the voltage deviates from the control range.

### Regulation Minimum Voltage

The minimum acceptable voltage at the regulated bus.

### Regulation Maximum Voltage

The maximum acceptable voltage at the regulated bus.

### Regulation Target Type

As long as the regulated voltage is inside the regulation minimum and maximum, then the transformer will not change its tap ratio. When the regulated voltage moves outside of this regulation range, then Simulator will calculate a new tap ratio in an attempt to bring the regulated voltage back inside of its range. The **Regulation Target Type** determines what value is used as a target when calculating this change in tap ratio. **Middle** is the default and means that the target is the average of the regulation minimum and maximum regardless of whether the voltage is high or low. **Max/Min** means that the regulation maximum is used as the target when the regulated voltage is above the maximum, and regulation minimum is used as the target value when the regulated voltage is below the minimum.

### Present Tap Ratio

The tap ratio of the transformer for the current system state.

### Minimum Tap Ratio, Maximum Tap Ratio

Minimum and maximum allowable off-nominal tap ratios for the LTC transformer. Typical values are 0.9 and 1.1.

### Tap Step Size

Transformer off-nominal turns ratio increment. The off-nominal turns ratio is either incremented or decremented from 1.0 in integer multiples of this value. Default value is 0.00625.

### Voltage to Tap Sensitivity

Shows the sensitivity of the voltage magnitude at the regulated bus to a change in the transformer's tap ratio. You can use this field to assess whether or not the transformer can effectively control the regulated bus voltage. In an ideal case, such as when the LTC transformer is being used to control the voltage at a radial load bus, the sensitivity is close to 1.0 (or -1.0 depending upon whether the tapped side of the transformer is on the load side or opposite side of the transformer). However, sometimes the transformer is very ineffective in controlling the voltage. This is indicated by the absolute value of the sensitivity approaching 0. A common example is a generator step-up transformer trying to control its high-side voltage when the generator is off-line. Simulator automatically disables transformer control if the transformer sensitivity is below the value specified on *Power Flow Solution Tab* of the [Simulator Options dialog](#).

### Impedance Correction Table

This field specifies the number of the transformer's corresponding transformer impedance correction table. Transformer impedance correction tables are used to specify how the impedance of the transformer should change with the off-nominal turns ratio. If this number is 0, then no impedance correction table is associated with the transformer, and the impedance of the transformer will thus remain fixed as the tap ratio changes. Valid impedance correction table numbers range from 1 to 63. To assign an existing impedance correction table to the transformer, enter the existing table's number. To view the existing impedance correction tables, click the *Insert/View Impedance Correction Table* button, which brings up the [Transformer Impedance Correction Dialog](#). To define a brand new impedance correction table for the transformer, enter an unused table number and then click *Insert/View Impedance Correction Table* to prescribe the correction table. Note that the association between a transformer and an impedance correction table is not finalized until you select either *OK* or *Save* on the [Line/Transformer Dialog](#).

### View Transformer Correction Table or Insert Transformer Correction Table

Click on this button either to view or to insert transformer correction tables. Clicking on this button displays the [Transformer Impedance Correction Dialog](#). Note that the table must prescribe at least two points in order to be defined.



---

## Transformer Mvar Control Dialog

---

The Transformer Mvar Control dialog is used to view the control parameters associated with load-tap-changing (LTC) transformers that are used to control the Mvar flow through the transformer. To view this display, click on the **Automatic Control Options** button on the [Line/Transformer Information Dialog](#).

When used to control reactive power, the LTC transformer always controls the reactive power flow at the *from* end of the transformer (i.e., the tapped side), with positive flow assumed to be going through the transformer to the *to* bus. Therefore the regulated bus field is not used.

The dialog has the following fields:

### Mvar Flow at *From* Bus

The current Mvar flow as measured at the *from* end of the line. This is the parameter the transformer tries to control.

### Mvar Error

If the Mvar flow at the *from* end violates the limits defined by the *Minimum Mvar Flow* and *Maximum Mvar Flow* fields, the *Mvar Error* field indicates by how much the flow falls outside the control range.

### Regulation Minimum Mvar Flow, Regulation Maximum Mvar Flow

Minimum and maximum allowable reactive power flow as measured at the *from bus*. The transformer attempts to regulate the reactive flow to fall within this range.

### Regulation Target Type

As long as the regulated Mvar flow is inside the regulation minimum and maximum, then the transformer will not change its tap ratio. When the regulated Mvar flow moves outside of this regulation range, then Simulator will calculate a new tap ratio in an attempt to bring the regulated Mvar flow back inside of its range. The **Regulation Target Type** determines what value is used as a target when calculating this change in tap ratio. **Middle** is the default and means that the target is the average of the regulation minimum and maximum regardless of whether the Mvar flow is high or low. **Max/Min** means that the regulation maximum is used as the target when the regulated Mvar flow is above the maximum, and regulation minimum is used as the target value when the regulated Mvar flow is below the minimum.

### Present Tap Ratio

The transformer's present off-nominal turns ratio.

### Minimum Tap Ratio, Maximum Tap Ratio

Minimum and maximum allowable off-nominal tap ratios for the LTC transformer. Typical values are 0.9 and 1.1.

### Tap Step Size

Transformer off-nominal turns ratio increment. The off-nominal turns ratio is either incremented or decremented from 1.0 in integer multiples of this value. Default value is 0.00625.

### Mvar to Tap Sensitivity

The amount of Mvar shift that would be implemented by switching one tap position from the current position. This sensitivity indicates the ability of the transformer to control Mvars.

### Impedance Correction Table

Specifies the number of the transformer's corresponding transformer impedance correction table. Transformer impedance correction tables are used to specify how the impedance of the transformer should change with the off-nominal turns ratio. If this number is 0, no impedance correction table is associated with the transformer, and the impedance of the transformer will thus remain fixed as the tap ratio changes. Valid impedance correction table numbers range from 1 to 63. To assign an existing impedance correction table to the transformer, enter the existing table's number. To view the existing impedance correction tables, click the *Insert/View Impedance Correction Table* button, which brings up the [Transformer Impedance Correction Dialog](#). To define a brand new impedance correction table for the transformer, enter an unused table number and then click *Insert/View Impedance Correction Table* to prescribe the correction table. Note that the association between a transformer and an impedance correction table is not finalized until you select either *OK* or *Save* on the [Line/Transformer Dialog](#).

#### **View Transformer Correction Table or Insert Transformer Correction Table**

Click on this button either to view or to insert transformer correction tables. Clicking on this button displays the [Transformer Impedance Correction Dialog](#). Note that the table must prescribe at least two points in order to be defined.

---

## Transformer Phase Shifting Information

---

The Transformer Phase Shifting Dialog is used to view the control parameters of phase-shifting transformers. To view this display, click on the **Automatic Control Options** button on the [Line/Transformer Information Dialog](#), provided that the *Phase Shift Control* option is chosen from the *Automatic Control* group.

### Regulated Bus Number

Number of the terminal bus of the phase shifter regulated by the phase shifter. When control is active, the phase shifter will automatically change its phase shift to keep the MW flow at this bus at the desired value.

### Current MW Flow

Current MW flow through the transformer measured at the regulated bus terminal.

### MW Error

If the current MW flow falls outside the minimum/maximum MW flow limits, the *MW Error* field indicates by how much the flow violates the regulating range.

### Regulation Minimum MW Flow, Regulation Maximum MW Flow

Minimum and maximum allowable MW flow through the phase shifter.

### Regulation Target Type

As long as the regulated MW flow is inside the regulation minimum and maximum, then the transformer will not change its tap ratio. When the regulated MW flow moves outside of this regulation range, then Simulator will calculate a new tap ratio in an attempt to bring the regulated MW flow back inside of its range. The **Regulation Target Type** determines what value is used as a target when calculating this change in tap ratio. **Middle** is the default and means that the target is the average of the regulation minimum and maximum regardless of whether the MW flow is high or low. **Max/Min** means that the regulation maximum is used as the target when the regulated MW flow is above the maximum, and regulation minimum is used as the target value when the regulated MW flow is below the minimum.

### Present Phase Angle (Degrees)

The phase angle of the transformer for the current solved system state.

### Minimum Phase Angle, Maximum Phase Angle

Minimum and maximum allowable phase shift in degrees.

### Step Size (Degrees)

Phase shift change per step in degrees.

### MW Flow to Phase Sensitivity

The sensitivity of the controlled MW flow to changes in the transformer's phase. This sensitivity indicates the transformer's ability to regulate its MW flow.

### Impedance Correction Table

Specifies the number of the transformer's corresponding transformer impedance correction table. Transformer impedance correction tables are used to specify how the impedance of the transformer should change with the off-nominal turns ratio. If this number is 0, no impedance correction table is associated with the transformer, and the impedance of the transformer will thus remain fixed as the tap ratio changes. Valid impedance correction table numbers range from 1 to 63. To assign an existing impedance correction table to the

transformer, enter the existing table's number. To view the existing impedance correction tables, click the *Insert/View Impedance Correction Table* button, which brings up the [Transformer Impedance Correction Dialog](#). To define a brand new impedance correction table for the transformer, enter an unused table number and then click *Insert/View Impedance Correction Table* to prescribe the correction table. Note that the association between a transformer and an impedance correction table is not finalized until you select either *OK* or *Save* on the [Line/Transformer Dialog](#).

#### **View Transformer Correction Table or Insert Transformer Correction Table**

Click on this button either to view or to insert transformer correction tables. Clicking on this button displays the [Transformer Impedance Correction Dialog](#). Note that the table must prescribe at least two points in order to be defined.

---

## Transformer Field Options Dialog

---

Transformer field objects are used to show field values specific to transformers. Use [Line Fields](#) to show fields generic to transformers and transmission lines, such as the flow of power through the device. The transformer fields dialog is used to view and modify the parameters associated with transformer-specific fields.

### Near Bus Number

Bus associated with the *near end* of the transformer.

### Far Bus Number

Bus associated with the *far end* of the transformer.

### Circuit

Two-character identifier used to distinguish between transformers joining the same two buses. Default is '1'.

### Find...

If you do not know the exact transformer you are looking for, you can click this button to open the [advanced search engine](#).

### Total Digits in Field

Total number of digits to show in the field.

### Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

### Delta per Mouse Click

This value is used only with the **Off-nominal Tap Ratio** and **Phase Shift Angle** field types. When there is a nonzero entry in this field, and the field type is valid, a spin button is shown to the right of the zone field. When the up spin button is clicked, the field value is increased by this number; when the down button is clicked, the field value is decreased by this amount.

### Field Value

Shows the current output for the transformer field. Whenever you change the **Type of Field** selection, this field is updated.

### Field Prefix

A prefix that can be specified and displayed with the selected value.

### Rotation Angle in Degrees

The angle at which the text is to appear on the oneline diagram.

### Anchored

When checked, the text field will move with the transformer if the transformer is moved on the oneline diagram.

### Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

### Type of Field

Designates the type of transformer field to show. The following choices are available:

**Off-nominal Tap Ratio** Actual tap ratio

**Phase Shift Angle** Actual phase shift in degrees

**Off-nominal Tap Position** Tap position in steps, usually ranging from L16 to R16

**Automatic Control Status** The status of the control for the transformer

Select **OK** to save changes and close the dialog or **Cancel** to close dialog without saving your changes.

---

### Three Winding Transformer Information

---

The Three Winding Transformer Dialog is used to create, modify or delete three winding transformer records in Edit Mode, or to view information for a specific three winding transformer record in run mode. Note that all the values displayed on this dialog are on the system MVA base. If the records were created from a file, the values are automatically converted to the system base. If a three winding transformer record is entered manually, the parameters need to be entered into Simulator computed with the same system MVA base Simulator is using.

This dialog has the following controls:

#### Primary Winding

This section of the dialog displays the primary winding terminal bus number, nominal kV, and fixed tap value (in per unit). In addition, the automatic tap changer is assumed to be on the primary winding of a three winding transformer. Therefore the LTC field displays the tap changer tap value (in per unit) on the primary winding.

#### Secondary Winding

This section of the dialog displays the secondary winding terminal bus number, nominal kV, and fixed tap value (in per unit).

#### Tertiary Winding

This section of the dialog displays the tertiary winding terminal bus number, nominal kV, and fixed tap value (in per unit).

#### Star Bus (Internal Node)

This section of the dialog displays the internal node parameters of the three winding transformer model. Three winding transformers are modeled as three two winding transformers connected at the three winding transformer terminal buses to a common or internal node, referred to as the star bus. The parameters displayed for the star bus are the bus number, voltage (in per unit), and angle.

#### Primary-Secondary, Secondary-Tertiary, and Tertiary-Primary Impedance

These are the actual three winding transformer winding to winding impedances, in per unit on the system base. These values are used to compute the equivalent two winding transformer impedances for the two winding transformers used to model the three winding transformer operation.

#### Transformer Parameters

The values for the magnetizing conductance (G) and magnetizing susceptance (G), in per unit on the system base.

#### Circuit ID

The circuit identifier for the three winding transformer.

#### Status

The status of the three winding transformer. If checked, the three winding transformer model is in service, otherwise the equivalent model is treated as out of service.

#### Mathematically equivalent two-winding transformers

This table displays the three two winding transformers that are mathematically equivalent representations of the three winding transformer. If you read the three winding transformer

record from a file, the two winding equivalent transformers are created automatically. If you are inserting a three winding transformer manually, you can set the parameters for the primary, secondary and tertiary windings in the fields above, then click the **Set Two-Winding Equivalent Transformers** button to have Simulator automatically create the two winding transformer records for you. You can also right-click in this table and insert, modify or delete two winding transformers manually if you already have the two winding transformer representations created.

Once you are finished with the dialog, you can click **Save** or **OK** to save any changes. If you wish to abandon any changes you have made, click **Cancel**.



---

**Switched Shunt Information (Edit Mode)**


---

This dialog is used to view and modify the parameters associated with each switched shunt in the system. It can also be used to insert new switched shunts and sometimes to delete existing shunts. Only one switched shunt is permitted at each bus. Switched shunts usually consist of either capacitors to supply reactive power (in MVR) to the system, or reactors to absorb reactive power. The switched shunts are represented by a number of blocks of admittance that can be switched in a number of discrete steps. If at least one block is in service, the shunt is said to be online. The shunt's corresponding circuit breaker can be used to determine and to toggle the switched shunt's status.

The Edit Mode version of this dialog is very similar in content to its [Run Mode](#) counterpart.

**Bus Number**

Unique number between 1 and 2,147,483,647 (equals  $2^{31}$  minus 1) used to identify the bus to which the switched shunt is attached. You can use the spin button immediately to the right of the number to move to the next switched shunt (click the up arrow) or the previous switched shunt (click the down arrow).

**Find By Number**

To find a switched shunt by its bus number, enter the number into the Bus Number field. Then click this button.

**Bus Name**

Unique alphabetic identifier for the bus to which the switched shunt is attached, consisting of up to eight characters.

**Find By Name**

To find a switched shunt by its bus name, enter the bus name into the Bus Name field (case insensitive). Then click this button.

**Shunt ID**

Since multiple switched shunts are allowed on a single bus, each switched shunt has a unique Shunt ID.

**Find...**

If you do not know the exact switched shunt bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

**Status**

Open or closed status of the switched shunt.

**Labels**

Clicking on this button will open the [Subscribed Aliases dialog](#) listing all the labels or aliases assigned for the selected load.

**Area Number, Area Name**

The area number and name to which the switched shunt belongs. Note that you can change the area of the switched shunt to be different than the area of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the switched shunt within a different area than that of the bus to which it is electrically connected.

**Zone Number, Zone Name**

The zone number and name to which the switched shunt belongs. Note that you can change the zone of the switched shunt to be different than the zone of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the switched shunt within a different zone than that of the bus to which it is electrically connected.

**Substation Number, Substation Name**

The name and number of the substation to which the switched shunt belongs, and it is the same than that of the terminal bus.

**Display**

**Display Size**

Size of the switched shunt.

**Scale Width with Size**

Automatically scales the width of the symbol when the object is resized.

**Display Width**

Width of the switched shunt symbol.

**Pixel Thickness**

Thickness of the display object in pixels.

**Orientation**

Specifies the direction in which to draw the object.

**Anchored**

If checked, the object is anchored to its terminal bus. See [Anchored Objects](#) for details.

**Link to New Shunt**

Adds a new record in the data or links the selected shunt to a different record.

**OK, Save, Delete, and Cancel**

**OK** saves your changes and closes the dialog. **Save** saves your changes but does not close the dialog; this allows you to use, for example, the Find By Number command to edit additional switched shunts. **Delete** deletes the current switched shunt; this option is not available when inserting objects graphically – use the cut command instead. **Cancel** closes the dialog but does not save any changes.

**Parameters**

**Nominal Mvar**

The Nominal Mvar field gives the initial amount of reactive power the device would supply (in Mvars) if its terminal voltage were 1.0 per unit.

**Nominal MW**

This field is only visible when a switched shunt object has been read from a file as a Bus Shunt. In that case, it is possible for the bus shunt to have both a MW and MVAR component. The MW component will be displayed here. In general, switched shunts of other control types do not have MW components, and this field will not be displayed. If you change a switched shunt read as a bus shunt to another form of control, the MW component will remain, but has no controllability.

**Control Mode**

Determines whether the switched shunt has a fixed value, or whether the amount of reactive power supplied by the device changes in either discrete steps or continuously in order to maintain its terminal voltage within the voltage range specified in the Voltage Regulation fields. This field can be changed (except in Viewer). However, for a switched shunt to be used for automatic control, three fields must be set correctly: 1) the *Control Mode* field must be set to either *Discrete* or *Continuous*, 2) the corresponding area's *Auto Shunts* property must be true, and 3) the case-wide *Disable Switched Shunt Control* option, which can be set on the *Power Flow Solution Tab* of the [Simulator Options Dialog](#), must not be checked.

**Note: automatic control of switched shunts is disabled if the voltage regulation high value is not greater than the low value; they should not be equal unless in the continuous mode.**

Note the additional control mode called Bus Shunt (Fixed). This is analogous to the shunt MW and MVAR values that can also be stored at the bus level. The difference is that bus shunts stored directly with the bus cannot be turned on and off in the load flow; rather they are always included in the load flow solution. Bus shunts that are represented as switched shunt objects on Bus Shunt control, however, are mathematically exactly the same and can be turned on or off. The reason for the differentiation of the Bus Shunt versus normal Fixed control is that the Bus Shunt control type is intended to identify the difference between a bus shunt and a transformer that MAY have controllability, but is currently turned off control by being set to a Fixed value.

## Control Regulation Settings

### Voltage Regulation

When the switched shunt is on automatic control, its reactive power is changed in discrete steps or continuously to keep the voltage at the regulated bus within the per unit voltage range defined by **High Value** and the **Low Value**.

In the case of discrete control, the amount of reactive power supplied by this device changes in discrete amounts, thus the High Value should be greater than the low value. The necessary voltage range depends upon the size of the switched shunt blocks. In addition to a voltage range for discrete control, a specific **Target Voltage** can be specified as well. The target voltage will try to be met, either approximately under discrete control, or exactly under continuous control (either true continuous or discrete with a continuous shunt correction element.) The number of the regulated bus is shown in the **Reg. Bus #** field.

### Generator Mvar Regulation

This option allows switched shunts to control generator Mvar output to better enable full var range inside the [inner power flow loop](#). During the first inner power flow loop, generator var limit checking is disabled for generators whose Mvar output is controlled by switched shunts. This will enable the use of the full reactive range of the switched shunts before generators hit their reactive limits. This corrects a problem in which the power flow fails to solve because generators are at their Mvar limits even though there is still reactive support available from switched shunts.

## Switched Shunt Blocks

The amount of shunt reactive power (susceptance) is specified in the Switched Shunt Block field. The columns in this field correspond to different blocks of reactive power. The first row indicates the number of steps in each block, and the second row gives the amount of nominal Mvars per step (assuming 1.0 per unit voltage). You may model both capacitors and reactors. The reactors should be specified first, in the order in which they are switched in, followed by the capacitors, again in the order they are switched in. The sign convention is

such that capacitors are positive and reactors negative. Shunt blocks are switched in order from left to right.

### Control Parameters

#### Single Largest Step

This option only applies when a switched shunt is set on discrete control. If checked the switched shunt will switch in EITHER all of the available reactor blocks OR all of the capacitor blocks at once when the voltage falls outside the given range. Whether the reactor or capacitor blocks switch is determined by which limit is violated. A switched shunt with this option checked will only switch ONCE during a load flow solution, and then remains fixed at the new output for the remainder of the same solution calculation.

#### Allow switching in the inner power flow loop

This option is available for individual discrete shunts only. If this option is set, discrete shunts are treated as continuous in the [inner power flow loop](#). This means that they are treated as PV buses in the inner power flow loop. After the first inner power flow loop, the shunt nominal Mvar setting is rounded to positive infinity to the next discrete step. If any shunts exist that are allowed to switch in the inner power flow loop, the inner power flow loop is repeated again with the shunts being treated as discrete in the subsequent inner power flow loop. Shunts that are allowed to switch in the inner power flow loop will only switch if the global option to "Disable Treating Continuous SSs as PV Buses" is not checked.

#### Use Continuous Element

If this option is checked, then Simulator will use a continuous element to fine-tune a discrete controlled switched shunt by injecting or absorbing additional MVARs to try and obtain the target voltage of the controlled bus.

#### Minimum and Maximum Susceptance

The minimum and maximum susceptance range for the continuous correction element.

#### Use High Target Voltage

Check this box to use the target voltage specified in the **High Target Value** edit box when the regulated point goes above the High limit. This will give a different target value if the voltage goes out of range on the high end than the low end. If the voltage goes out of range on the low end, the original target value on the Parameters page will be used. If this box is unchecked, only the target value on the parameters page will be used, whether the violation is high or low.

### Fault Parameters

Typically switched shunts are treated as open circuits in the zero sequence data for fault analysis. However, it is possible to define zero sequence admittance blocks to be used. The blocks work similarly to the load flow Switched Shunt Blocks discussed above. Usually there will be the same number of blocks in the zero sequence data as in the load flow data. Simulator will determine how many blocks were switched in for the power flow solution, and then use the zero sequence block data to calculate the zero sequence admittance for the same number of blocks.

### Custom

This page of the dialog can be used to enter notes about the switched shunt. Any information entered in the memo box will be stored with the case when the case is saved to a PWB file. Custom fields can also be viewed and added from this page as well.

---

## Switched Shunt Field Information

---

Switched shunt field objects are used primarily to indicate various quantities associated with switched shunt devices. Furthermore, some switched shunt field types, which are distinguished by an integrated spin button, may be used to change switched shunt device properties.

The Switched Shunt Fields Information Dialog can be used to modify the properties of individual switched shunt fields on the oneline. The dialog displays the following fields:

### Find...

If you do not know the exact bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

### Bus Number

Number of the bus to which the switched shunt associated with the field is connected. Use the dropdown box to view a list of all buses with switched shunts in the case with [valid area/zone/owner filters](#).

### Bus Name

Name of the bus to which the switched shunt associated with the field is connected. Use the dropdown box to view a list of all buses with switched shunts in the case with [valid area/zone/owner filters](#).

### ID

ID of the switched shunt associated with the field.

### Total Digits in Field

Total number of digits to show in the field.

### Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

### Field Value

The current value of the field being displayed.

### Field Prefix

A prefix that can be specified and displayed with the selected value.

### Delta Per Mouse Click

Switched shunt fields can be used not only to show various fields associated with switched shunt devices, but they can also be used to change some values. This is accomplished using spin buttons shown to the right of the switched shunt field. When the up spin button is clicked, the switched shunt field value is increased by the amount specified in the *delta per mouse click* field. When the down spin button is clicked, the switched shunt field value is decreased by the same amount.

This field is only used for Switched Shunt Mvar fields. Specifying a nonzero value in this field causes the integrated spin button to appear as part of the switched shunt field on the oneline.

### Rotation Angle in Degrees

The angle at which the text is placed on the diagram, in degrees.

**Anchored**

If the *Anchored* checkbox is checked, the switched shunt field is [anchored](#) to its associated switched shunt, which means that it will move with the switched shunt.

**Include Suffix**

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

**Type of Field**

Used to determine the type of switched shunt field to show. The following choices are available:

**Switched Shunt Mvar** Total Mvar capacitance at the bus

**Select a Field** Choose from any of the different switched shunt fields

Select **OK** to save changes and to close the dialog, or click **Cancel** to close the dialog without saving your changes.

---

**Bus Information (Run Mode)**


---

This dialog is used to view information about each bus in the system. It can be displayed by right-clicking on any bus on the online and choosing **Bus Information Dialog** or choosing **Show Dialog** from the [Bus Display](#). This dialog can only be reached in Run Mode, but has a similar [Edit Mode](#) counterpart. The Bus Information Dialog has the following fields:

**Bus Number**

Unique number between 1 and 2,147,483,647 (equals  $2^{31}$  minus 1) used to identify the bus. You can use the small arrow immediately to the right of the number to view a list of all buses in the case with valid [display filters](#). Or you can use the spin button further to the right of the number to move to the next bus (click the up arrow) or the previous bus (click the down arrow).

**Find By Number**

To find a bus by its number, enter the number into the **Bus Number** field and then click this button.

**Bus Name**

Unique alphabetic identifier for the bus. You can use the small arrow immediately to the right of the bus name to view a list of all bus names in the case with [valid display filters](#).

**Find By Name**

To find a bus by its name, enter the bus name into the **Bus Name** field (case insensitive) and then click this button.

**Find...**

If you do not know the exact bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

**Nom. Voltage**

Nominal voltage of the bus.

**Labels**

Clicking on this button will open the [Label Manager dialog](#) listing all of the labels assigned for the selected bus.

**Area Number, Name**

Each bus is associated with an Area record. These fields show the number and name of this area. See [Area Records Display](#) for more details about areas.

**Zone Number, Name**

Each bus is associated with a Zone record. These fields show the number and name of the zone. See [Zone Records Display](#) for more details about zones. You can also use the [Zone Dialog](#) to list the buses in a particular zone and to easily move a group of buses from one zone to another.

**Substation Number, Name**

Each bus can be associated with a Substation record. By default, buses are not assigned to substations, and in that case these fields are blank. See the [Substation Records Display](#) topic for more details on adding substations.

**Owner Number, Name**



Each bus is associated with an Owner record. These fields show the number and name of the owner. See [Owner Records Display](#) for more information about Owners.

#### **Voltage (per unit)**

Bus voltage in per unit notation. You may enter a new per unit voltage magnitude. However, the only effect this has is changing the initial voltage guess used in the iterative solution. If you would like to change the reference voltage for a generator, please see [Generator Information Dialog](#).

#### **Voltage (kV)**

Bus voltage in actual kilovolts.

#### **Angle (deg) and Angle (rad)**

Voltage angle at the bus in degrees and radians. You may enter a new voltage angle. However, the only effect this has is changing the initial voltage guess used in the iterative solution EXCEPT AT THE SLACK BUS. Changing the angle for the slack bus will shift the voltage angle for all the buses in the slack bus' island by a similar amount.

#### **Status**

Status of the bus, either connected or disconnected. A disconnected bus is not energized. You can use this field to change the status of the bus. When the bus is initially connected, selecting **Disconnected** opens all of the transmission lines incident to the bus, disconnecting the bus from the rest of the system. Selecting **Connected** closes all of the lines incident to the bus unless they attach to another disconnected bus.

#### **View Substation Dialog**

Clicking on this button will open the [substation dialog](#) for the substation the bus is contained in.

#### **View Owner Dialog**

Clicking on this button will open the bus' [owner dialog](#).

#### **Slack Bus**

Checked only if the bus is a system slack bus. This value can only be changed in the Edit Mode.

#### **View All Flows at Bus**

Clicking on this button will open a [quick power flow list](#) for the current bus.

### **Device Info**

#### **Generator Information**

Displays the total MW and Mvar generation at the bus. You cannot change either of these fields from this display. Select **View/Edit Generator Records** to view the individual generator records for the bus. Selecting this button displays the [Generator Dialog](#) for the first generator at the bus.

#### **Load Information**

Displays the total MW and Mvar load at the bus. You cannot change either of these fields from this display. Select **View/Edit Bus Load Records** to view the individual load records for the bus. Selecting this button displays the [Load Dialog](#).

#### **Shunt Admittance**

Shows the real and reactive components of the shunt admittance to ground. Entered in either MW or Mvar, assuming one per unit voltage. B is positive for a capacitor and negative for a reactor. If B corresponds to a switched device, consider using a [switched shunt](#).

### **Bus Voltage Regulation**

This section lists the devices (if any) that are controlling the voltage at the bus.

### **Desired PU Voltage**

If the bus is being regulated by one or more devices, this field will display the desired regulated voltage (in per unit) the devices are attempting to maintain.

### **Fault Analysis Load Parameters**

This page of the display is only available for buses which have one or more load attached to the bus. The parameters on this tab are used when running a fault analysis study. The values represent the total load at the bus for the negative and zero sequence as equivalent admittances. By default, these values are zero. For load buses, these values can be changed by the user, or they can be specified by loading short circuit data from within the [Fault Analysis Dialog](#). It is also possible to define these values as non-zero at a bus where no load exists in the load flow, but it is not usually desirable to do so.

### **OPF**

This tab is only available if you have the Optimal Power Flow (OPF) add-on tool for PowerWorld Simulator. This tab displays the MW marginal cost (Locational Marginal Price) for the bus when performing an OPF solution. The page also breaks down the LMP into its cost components.

### **Geography**

The Geography page of the dialog provides information on the geographic location of the bus on the currently active oneline diagram.

### **Custom**

The Custom page of the dialog contains two sections: custom fields and memo.

The custom fields section allows access to setting and changing the values for custom fields that have been defined for the bus. Defining custom fields is detailed in [Custom Field Descriptions](#).

The [Memo section](#) of the dialog is simply a location to log information about the bus. Any information entered in the memo box will be stored with the case when the case is saved to a [PWB](#) file.

---

## Substation Information (Run Mode)

---

This dialog is used in the Run Mode to view and modify information associated with a substation record. It displays different information from the [Edit Mode](#) version of the [substation dialog](#). To display it from [Run Mode](#), first select **Aggregation > Substations** in the [Model Explorer](#) to bring up the [Substation Records Display](#). Right-click on the substation of interest and choose **Show Dialog**. The Run Mode Substation Dialog has the following fields:

### Substation Number

An integer identifier for the substation. You can use the spin button immediately to the right of this field to move to either the next substation (click the up arrow) or the previous substation (click the down arrow).

### Substation Name and ID

Two alphanumeric identifiers for the substation.

### Find By Number

To find a substation by its number, enter the number into the **Substation Number** field, then click this button.

### Find By Name

To find a substation by its name, enter the name into the **Substation Name** field, then click this button.

### Find By Sub ID

To find a substation by its substation ID, enter the ID into the **Substation ID** field, then click this button.

### Find...

If the exact substation number, name and ID are not known, you can use the [Find Dialog](#) to search for and select a substation from a list of substations.

### Labels

Clicking this button will open a dialog displaying the list of defined labels for the substation. New labels can also be added for the substation from the dialog as well.

### View All Flows at Substation

Clicking this button will open a [quick power flow display](#) listing the buses contained in the substation.

## Information

### Load and Generation

Real and reactive load, generation, shunts, losses, and interchange for the substation.

### Bus Voltages

Summary information on all buses in the substation, including total number of buses, number of dead (disconnected) buses, and minimum and maximum bus voltage and angle within the substation.

### Available Gen MW/Mvar Ranges

Total amount of generation increase or decrease available for all generators in the substation.

### **Buses**

The Buses table identifies the buses in the substation, and provides summary information on each.

### **Gens**

The Gens table identifies the generators in the substation, and provides summary information on each.

### **Loads**

The Loads table identifies the loads in the substation, and provides summary information on each.

### **Switched Shunts**

The Switched Shunts table identifies the switched shunts in the substation, and provides summary information on each.

### **Tie Lines**

The Tie Line Table identifies the flows on all of the substation's ties to other substation.

### **Custom**

This page of the dialog can be used to enter notes about the substation. Any information entered in the memo box will be stored with the case when the case is saved to a PWB file. Custom fields can also be entered for storage with the substation object viewed.

### **Geography**

Displays geographic information about the location of the substation, in Latitude and Longitude.

---

## Generator Information (Run Mode)

---

This dialog is used to view information about each generator in the system. Many fields on this display can also be changed (except in Viewer). Here we describe the Run Mode version of the Generator Information Dialog. The [Edit Mode](#) version is very similar.

### Bus Number

Unique number between 1 and 99,999 used to identify the bus to which the generator is attached. The dropdown list enumerates all generator buses in the case that meet the criteria established by [display filters](#). You may select a bus number directly from the dropdown list, or you may use the spin buttons to cycle through the list of generator buses.

### Bus Name

Unique alphabetic identifier for the bus to which the generator is attached, consisting of up to eight characters. Use this dropdown box to view a list of all generator bus names in the case with [valid display filters](#).

### ID

Two character alphanumeric ID used to distinguish multiple generators at a bus; '1' by default.

### Find By Number

To find a generator by its number and ID, enter the number into the **Bus Number** field and the ID into the **ID** field. Then click the **Find By Number** button.

### Find By Name

To find a bus by its name and ID, enter the bus name into the **Bus Name** field (case insensitive) and the ID into the **ID** field. Then click the **Find By Name** button.

### Find...

If you do not know the exact generator bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

### Status

Status of the generator, either Closed (connected to terminal bus) or Open (not connected). You can use this field to change the status of the generator.

### Area Name

Name of the area in which the generator's terminal bus is located.

### Labels

Clicking on this button will open the [Subscribed Aliases dialog](#) listing all the labels or aliases assigned for the selected generator.

### Same Owner as Terminal Bus

Read-only check-box that indicates whether the generator's owner is the same than the terminal bus' owner.

### Fuel Type

Type of fuel used by the generator this model represents. In most cases, this field is unnecessary for normal load flow analysis, and hence the default value is Unknown. However, this value can be useful during the [Security Constrained OPF](#) analysis.

### Unit Type

The type of unit the generator represents, such as combined cycle, steam, hydro, etc.

There additional sections of generator information available from the Run Mode generator dialog:

[Power and Voltage Control](#)

[Generator Cost Information](#)

[OPF](#)

[Fault Parameters](#)

[Owners, Area, Zone, Sub](#)

[Custom](#)

---

## Generator Information: Power and Voltage Control

---

The **Power Control** grouping fields are used to show/change the values associated with the real power output of the generator.

### MW Output

Current real power output of the generator.

### Minimum and Maximum MW Output

Minimum and maximum real power output limits for the generator. Simulator will not let the MW output go below its minimum value or above its maximum value if the *Enforce MW Limits* option is exercised.

### Available for AGC

Determines whether or not the generator is available for automatic generation control (AGC). Normally this box should be checked. However, there are times when you would like to control the generator output manually (such as if you are using the generator to remove a line limit violation), in which case you should leave this box unchecked. A generator is also placed on "manual" control any time you manually change its output. You could then place the generator back on AGC control by using this dialog.

### Enforce MW Limits

If checked, the minimum and maximum MW limits are enforced for the generator, provided the **Enforce Generator MW Limits** field is also checked on the Limits Tab of the [Simulator Options Dialog](#). If this box is checked and a generator is violating a real power limit, the generator's MW output is immediately changed.

### Participation Factor

The participation factor is used to determine how the real power output of the generator changes in response to demand when the generator is available for AGC and the area is on [participation factor control](#). When you open a case using the [PTI Raw Data Format](#), this field is initialized to the per unit MVA rating of the generator, since participation factor information is not stored in the PTI format.

### Loss Sensitivity

Shows how the losses for an area will change for an incremental increase in the generation at the bus. This information is useful in determining the economic dispatch for the generation. The implicit assumption in calculating this field's value is that the incremental change in generation will be absorbed by the system "slack bus." This field cannot be changed.

The **Voltage Control** grouping is used to show/change values associated with controlling the voltage/reactive power output of the generator.

### Mvar Output

Current reactive power output of the generator. You can manually change this value only if **Available for AVR** is not checked.

### Min and Max Mvar Output

Specify the minimum and maximum allowable reactive power output of the generator.

### **Available for AVR**

Designates whether or not the generator is available for automatic voltage regulation (AVR). When the AVR field is checked, the generator will automatically change its reactive power output to maintain the desired terminal voltage within the specified reactive power range. If a reactive limit is reached, the generator will no longer be able to maintain its voltage at the setpoint value, and its reactive power will then be held constant at the limit value.

### **Use Capability Curve**

If checked, the generator's reactive power limits are specified using a reactive capability curve that prescribes the dependence of the generator's reactive power limits on its real power output. Otherwise, the fixed values given in the **Min Mvar Output** and **Max Mvar Output** fields are used. The generator reactive capability can be defined using the table that appears at the bottom of the dialog. Please see [Generator Reactive Power Capability Curve](#) for details.

### **Regulated Bus Number**

Number of the bus whose voltage the generator is regulating. This is usually, but not always, the generator's terminal bus. Multiple generators can regulate the same remote bus, but the regulated bus must not be another generator bus. If the generator is at a slack bus, it must regulate its own terminal voltage. Select **Solution Details > Remotely Regulated Buses** from the [Model Explorer](#) to view the [Remotely Regulated Bus Records Dialog](#), which identifies all buses that are being remotely regulated.

### **Desired Reg. Bus Voltage**

Specifies the desired per unit voltage for the generator at the regulated bus. The regulated bus need not be the terminal bus of the generator.

### **Actual Reg. Bus Voltage**

Shows the actual per unit voltage at the regulated bus. If the generator is on AVR and has not reached a reactive power limit, the actual regulated bus voltage should be equal to the desired regulated bus voltage. This field cannot be changed.

### **Remote Reg %**

This field is only used when a number of generators at different buses are regulating a remote bus (i.e., not their terminal buses). This field then specifies the percentage of the total reactive power required by the remote bus to maintain its voltage that should be supplied by this generator. The default value is 100.



---

## Generator Options: Costs

---

The **Costs** tab of the Generator Information dialog (run mode) is used to show/change values associated with the cost of operation of the generator. See [Generator Cost Information](#) for details. Cost data can also be saved/loaded using the [Generator Cost Data files](#).

### Output Cost Model

#### Cost Model

Simulator can model generators as not having a cost model, or having either a cubic cost model or a piecewise linear model. The cost model type you choose determines the content of the remainder of this dialog

#### Unit Fuel Cost

The cost of fuel in \$/MBtu. This value can be specified only when you have chosen to use a cubic cost model.

#### Variable O&M

The Operations and Maintenance costs. Only used for cubic cost models.

#### Fixed Costs (costs at zero MW Output)

The fixed costs associated with operating the unit. These costs are independent of the generator's MW output level and is added to the cost prescribed by the cubic or piecewise linear model to obtain the total cost of operating the generator in \$/MWHr. The total fixed cost is the addition of the fuel cost independent value and the fuel cost dependent value multiplied by unit fuel cost.

#### Cubic Cost Coefficients A, B, C, D

For cubic cost models of the form  $C(P_{gi}) = (d \cdot P_{gi}^3 + c \cdot P_{gi}^2 + b \cdot P_{gi}) \cdot (\text{fuel cost}) + \text{fixed costs}$  (as described above), specify the cost curve's coefficients. The A coefficient is historically the fuel cost dependent fixed cost value, which is combined separately now with the fuel cost independent value. These coefficients can be specified only when you have chosen to use a cubic cost model.

#### Piecewise Linear Table

If you have chosen to use a piecewise linear cost model, a table appears that allows you to specify pairs of MW output levels and corresponding generator operating costs. To insert a new point on the cost curve, right-click on the table and choose *Insert New Point* from the resulting local menu. To delete an existing point from the cost curve, right-click on the table and choose *Delete Point* from the resulting local menu. To edit an existing point in the table, simply enter your changes to the appropriate cells.

#### Convert Cubic Cost to Linear

Use this option to create a piecewise linear cost function from the cubic cost function specified by the coefficients A, B, C, and D and the fuel cost. Specify the number of break points, and hence the number of segments, in the **Number of Break Points** field. Click the **Convert to Linear Cost** button to create the piecewise linear function that approximates the cubic cost function. This action switches **Cost Model** option to *Piecewise Linear* and displays the **Piecewise Linear Table** that identifies the piecewise linear curve's breakpoints.

### Bid Scale / Shift

#### Cost Shift, Cost Multiplier

The cost shift and cost multiplier allow you to easily apply a shift to the cost function for the purpose of assessing how variations in bids impact profit. The cost function is affected based on the following equation:

$$(\text{Original Cost Function} + \text{Cost Shift}) * \text{Cost Multiplier}$$

**Marginal Cost (run mode only)**

Shows the marginal cost of producing real power at the generator at its current output level,  $dC_i(P_{gi})/dP_{gi}$ .

**ED/OPF Cost (run mode only)**

This is the cost of production for this generator following an economic dispatch or optimal power flow solution, *including* the scaling from the cost shift and cost multiplier fields.

**Unscaled Cost (run mode only)**

The cost of production of the generator, *ignoring* the cost multiplier and cost shift. This cost is the result of the original cost function by itself.

---

**Generator Information: OPF**

---

The fields on this tab display information regarding the generator's participation in an OPF load flow solution.

**OPF MW Control**

The type of control the generator is allowed during an OPF solution. The generator can be set to No control during OPF, control only if its AGC property is set to Yes, or to always be controlled by the OPF regardless of the AGC status of the generator.

**Fast Start Generator**

The generator is being treated as a fast start generator during the OPF solution.

**Generator MW limits**

The MW limits of a generator can be altered in this location if you wish for the generator to use different limits than originally assigned in the load flow case, without actually changing the original values. Simply change the Current Min MW Limit and Current Max MW Limit to alter the limits observed by the generator during and OPF solution.

**MW Marginal Cost for Generator's Bus**

The OPF solved marginal cost at the generator's terminal bus.

**Initial, Final and Delta MW Output**

The MW output information for the generator resulting from the OPF run.

**Initial, Final and Delta Hourly Cost**

The hourly cost information for the generator resulting from the OPF run.

**OPF Results**

This section shows the results of OPF for the generator.

---

### **Generator Options: Owners, Area, Zone, Sub**

---

This information is located on the [Generator Information Dialog](#).

This tab is used to display or change the generator's owner information, area information, zone information, and substation information.

#### **Same Owner as Terminal Bus**

Read-only check-box that indicates whether the generator's owner is the same than the terminal buses' owner.

#### **Owners**

Currently, Simulator supports up to four owners for generators. To add an owner of a generator, change one of the Owner fields to a new owner number, and update the owner percentages accordingly. To modify an owner's percentage of ownership, simply modify the value in the percentage field for that owner. If you set the percentage of an owner to 0, that owner will be removed from the list of owners for the device. You can also remove an owner from owning part of a device by changing the owner field for that owner to 0. Note that if you do not set the new owner percentages of all specified owners such that the total is 100%, Simulator will normalize the percentages such that the total is 100% when you click **Save** or **OK** on the generator dialog.

#### **Area Number, Area Name**

The area number and name to which the generator belongs. Note that you can change the area of the generator to be different than the area of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the generator within a different area than that of the bus to which it is electrically connected.

#### **Zone Number, Zone Name**

The zone number and name to which the generator belongs. Note that you can change the zone of the generator to be different than the zone of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the generator within a different zone than that of the bus to which it is electrically connected.

#### **Substation Number, Substation Name**

The name and number of the substation to which the generator belongs, and it is the same than that of the terminal bus.

---

**Load Information (Run Mode)**


---

The Load Information Dialog can be used to inspect and modify the model of a bus load. To view the Load Information Dialog, select the load and choose **Show Dialog** from the [case information toolbar](#) or right-click on the load of interest and select **Load Information Dialog** from the resulting local menu. This is very similar to its [Edit Mode](#) counterpart. The dialog has the following fields:

**Bus Number**

Unique number between 1 and 2,147,483,647 (equals  $2^{31}$  minus 1) used to identify the bus to which the load is attached. The dropdown box provides a list of all load buses with [valid display filters](#). You can use the spin button to cycle through the list of load buses.

**Bus Name**

Unique alphabetic identifier for the bus to which the load is attached, consisting of up to eight characters. The dropdown box lists the names of all load buses in the case with [valid display filters](#).

**ID**

Two-character ID used to distinguish multiple loads at a bus. By default, the load id is equal to "1 ." An identifier of '99' is used to indicate an equivalent load.

**Find By Number**

To find a load by its number and ID, enter the number into the Bus Number field and the ID into the ID field. Then click this button.

**Find By Name**

To find a load by its name and ID, enter the bus name into the Bus Name field (case insensitive) and the ID into the ID field. Then click this button.

**Find...**

If you do not know the exact load bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

**Status**

Status of the load, either Closed (connected to terminal bus) or Open (not connected). You can use this status field to change the load's status.

**Labels**

Clicking on this button will open the [Label Manager](#) dialog listing all the labels assigned for the selected load.

**Area Number, Area Name**

Number and name of the area the load is a member of.

**Zone Number, Zone Name**

Number and name of the zone the load is a member of.

**Owner Number, Owner Name**

Number and name of the owner the load is a member of. Loads DO NOT have to be owned by the same owner as the terminal bus.

**Substation Number, Substation Name**

Number and name of the substation the load is a member of.

### **Load Information**

#### **Base Load Model, Current Load**

The Base Load Model fields are used to represent the amount of base real and reactive load at the bus. Usually this load is modeled as being "constant power," meaning that the amount of load is independent of the bus voltage magnitude. However, Simulator also permits modeling "constant current" load, for which the load varies in proportion to the bus voltage magnitude, and "constant impedance" load, for which the load varies in proportion to the square of the bus voltage magnitude. Values in these fields are specified in MW and MVR assuming one per unit voltage. All six fields in the Base Load Model section can be changed.

#### **Load Multiplier**

The actual load at the bus is equal to the base value multiplied by the corresponding load multiplier. The load multiplier is a value specifying how the load is scaled. The load multiplier depends upon the area load multiplier and the zone load multiplier. See [Load Modeling](#) for more details. The load multiplier value cannot be changed on this dialog.

#### **Bus Voltage Magnitude**

Voltage magnitude of the load's terminal bus.

### **OPF Load Dispatch**

The information on this tab displays the load information resulting from the load's participation in an OPF solution

#### **Benefit Model**

If this field is set to none, the load will not be dispatchable in the OPF solution. If the option is set to Piecewise Linear, the load is dispatchable during the OPF, according to the following fields.

#### **Min. and Max. MW Demand**

The minimum and maximum MW demand the load must operate within during the OPF solution.

#### **Available for AGC**

The load is only available for redispatch during the OPF solution if this option is checked.

#### **Fixed Benefit**

Value of the load benefit at minimum demand.

#### **Piece-wise Linear Benefit Curve**

This table the MW demand levels and their corresponding load benefit values, which in turn define the starting points and slopes of the piece-wise linear benefit curve segments.

### **Custom**

The Custom page of the dialog contains two sections: custom fields and memo.

The custom fields section allows access to setting and changing the values for custom fields that have been defined for the load. Defining custom fields is detailed in [Custom Field Descriptions](#).

The [Memo section](#) of the dialog is simply a location to log information about the load. Any information entered in the memo box will be stored with the case when the case is saved to a [PWB](#) file.

---

## Line/Transformer Information (Run Mode)

---

The Line/Transformer Information dialog box is used to view information about each transmission line and transformer in the system. You may use this dialog also to change many of the properties of lines and transformers (except in Viewer).

The Run Mode version of this dialog is very similar in content to its [Edit Mode](#) counterpart.

The Line/Transformer dialog sports the following fields:

### From Bus Number and Name

*From Bus* number and name. For transformers, the *from bus* is the tapped side.

### To Bus Number and Name

*To Bus* number and name.

### Circuit

Two-character identifier used to distinguish between multiple lines joining the same two buses. Default is '1'.

### Find By Number

To find a line or transformer by its bus numbers, enter the *from* and *to* bus numbers and the circuit identifier. Then click this button. Use the spin button to cycle through the list of lines and transformers in the system.

### Find By Name

To find a line or transformer by the names of its terminal buses, enter the *from* and *to* bus names and the circuit identifier. Then click this button.

### Find...

If you do not know the exact from and to bus numbers or names you are looking for, you can click this button to open the [advanced search engine](#).

### From End Metered

This field is only used for lines and transformers that serve as tie lines, which are lines that join two areas. If this field is checked for a tie line, then the *from* end of the device is designated as the metered end. Otherwise the *to* end is metered. By default, the *from* end is metered. The location of the metered end is important in dealing with energy transactions because it determines which party must account for transmission losses.

### From and To Bus Area Name

Names of the areas in which the From and To buses are located.

### From and To Bus Nominal kV

From and To bus nominal voltage levels.

### From and To Bus Voltage (p.u.)

The actual terminal bus voltages of the transmission element, in per unit.

### Labels

Clicking on this button will open the [Subscribed Aliases dialog](#) listing all the labels or aliases assigned for the selected branch.



## Parameters

### Status

Current status of the device.

### Per Unit Impedance Parameters

The resistance, reactance, the total charging susceptance (that is,  $B$ , not  $B/2$ ), and the total shunt conductance of the device (in per unit).

### Line Shunts

Select to view the [Line Shunt](#) Dialog. This dialog is used to only view the values of the line shunts in Run Mode. Line shunts are expressed in terms of the per-unit conductance and susceptance at each end of the line or transformer. If the line has shunts, the check box **Has Line Shunts** is checked.

### MVA Limits

Ratings for the transmission line or transformer in MVA. Simulator allows the use of up to eight different limits.

### Flows

These next fields show the actual real and reactive power flow at both ends of the device (because of real and reactive losses these numbers may be different), and its percentage MVA loading.

## Transformer Info

This fields on this tab are enabled only if the branch is a transformer. See the [Transformer Modeling](#) help for more information on transformer types and controls.

### Off-nominal Turns Ratio

The off-nominal tap ratio for the transformer.

### Phase Shift Degrees

Phase angle for the transformer. This field is usually non-zero only for phase shifting transformers or wye-delta and delta-wye connected transformers, but can be non-zero for an LTC or fixed transformer on rare occasions.

### Automatic Control Enabled

Check this box to enable automatic control of the transformer. Note that automatic control will be implemented only if (1) transformer control has been enabled for the transformer's area (see [Area Display](#) for details) and (2) transformer control has not been disabled for the entire case (via the *Disable Transformer Control* option on the *Power Flow Solution* tab of the [Simulator Options Dialog](#)). The Line/Transformer Dialog gives you convenient access to these control options through the *Case Control Enabled* and *Area Transformer Control Enabled* check boxes.

### Automatic Control Type

Depending on the type of control of the transformer, the **Change Automatic Control Options** button will open up the control options for [LTC Control](#), [MVAR Control](#), or [Phase Shifter Control](#).

### Area and Case Control Options

All these boxes must be checked for a transformer to be controlled. **Area Transformer Control Enabled** has to be checked for any transformer in that area to be controlled; **Case LTC Transformer Control Enabled** has to be checked for any transformer in the entire

case to be controlled, and **Case Phase Shifter Control Enabled** has to be checked for any phase shifter in the entire case to be controlled. Checking these does NOT automatically make the entire area or entire case on control.

### Series Capacitor

The parameters on this tab are enabled when the selected branch is a series capacitor. If the branch is a series capacitor, the **Is Series Capacitor** box will be checked. In addition, the **Status** field for series capacitors will be enabled, allowing you to change the Bypassed or In Service status of the series capacitor. The series capacitor status IS NOT the same as the branch status of Open or Closed.

### OPF

The OPF tab is only visible if you have the OPF ([Optimal Power Flow](#)) add-on tool for PowerWorld Simulator.

#### Enforce Line Flow Limit

This check box must be checked if the branch limit is going to be enforced when running an OPF solution. If this box is not checked, the OPF routine will allow the branch to violate its branch limits.

#### Treat Limit as Equality Constraint

If checked, the OPF solution will attempt to solve the load flow while keeping the flow on the branch at its limit.

#### Maximum MVA Flow

The largest MVA flow value measured on the line, either at the From or To bus.

#### Present MVA Limit

The limit enforced by the OPF for the branch. This is set in the [OPF constraint options](#), and is related to the original branch limits.

#### Maximum Percentage

The highest percentage of flow measured on the line, either at the From or To bus.

#### Limit Marginal Cost

The cost of enforcing the branch MVA limit.

#### Included in LP

Specifies whether or not the branch flow and limit was included as a constraint in the OPF solution. In general, branches that are not near their limit and do not appear to be changing flow dramatically towards their limit will be ignored in the OPF calculation to speed up the solution. **No** and **Yes** indicate whether or not the OPF process determined that the line needed to be included. The user can initially force the branch to be included or not included with these two fields. By choosing **Always**, the branch will be included in the OPF solution constraints regardless of the propensity of the line to be approaching its limit.

#### MVA Flow Constraint Status

The constraint status for the line in the OPF solution will be shown here with the corresponding boxes checked by Simulator. These check boxes cannot be changed manually by the user.

#### From/To Bus MW Marginal Costs

Displays the marginal costs of the branches terminal buses, following the solution of the OPF.

### Fault Info

The parameters on this tab are used when running a [fault analysis](#) study. The values represent the zero sequence impedance and zero sequence line shunt admittances for the analysis. By default, the positive and negative sequence line impedances and line shunt admittances are the same as the load flow impedance. The same fields are used for transformers, along with the configuration field. The configuration field defines the winding type combinations for the transformer (wye, delta, etc.) As a default, Simulator assumes a grounded wye to grounded wye transformer, which has the same model as a transmission line. Usually transformers are not of this type, and the proper type would need to be defined either manually or loaded from an external file in order for the fault analysis to be accurate.

### Owner, Area, Zone, Sub, PTDF

The **Default Owner (Same as From Bus)** read-only check-box indicates whether the line's owner is the same than the from bus' owner. Transmission elements can have up to four different owners, each with a certain owner percentage. To add an owner of a transmission element, change one of the Owner fields to a new owner number, and update the owner percentages accordingly. Note that if you do not set the new owner percentages of all specified owners such that the total is 100%, Simulator will normalize the percentages such that the total is 100% when you click **Save** or **OK** on the line/transformer dialog.

The area, zone and substation to which the From and To buses belong, are also shown. If a PTDF calculation has been performed, the PTDF values for the viewed line will be displayed as well.

### Custom

This page of the dialog can be used to enter notes about the transmission line. Any information entered in the memo box will be stored with the case when the case is saved to a PWB file. You can also enter custom strings and values that are also stored with the case.

---

## Transformer AVR Dialog

---

The Transformer AVR Dialog is used to view the control parameters associated with load-tap-changing (LTC) transformers when they are used to control bus voltage magnitudes. To view this display, click the *Automatic Control Options* button on the [Line/Transformer Information Dialog](#).

This dialog has the following fields:

### Automatic Control Type

This section is only visible on the dialog in edit mode. The control type of the transformer can be changed between no control, voltage regulation, reactive power control, or phase shifter control.

### Regulated Bus Number

The number of the bus whose voltage is regulated by the control.

### Present Regulated Bus Voltage

The present voltage of the regulated bus.

### Voltage Error

If the regulated bus' voltage falls outside the regulating range of the transformer (as defined by the *Minimum Voltage* and *Maximum Voltage* fields), the *Voltage Error* field indicates by how much the voltage deviates from the control range.

### Regulation Minimum Voltage

The minimum acceptable voltage at the regulated bus.

### Regulation Maximum Voltage

The maximum acceptable voltage at the regulated bus.

### Regulation Target Type

As long as the regulated voltage is inside the regulation minimum and maximum, then the transformer will not change its tap ratio. When the regulated voltage moves outside of this regulation range, then Simulator will calculate a new tap ratio in an attempt to bring the regulated voltage back inside of its range. The **Regulation Target Type** determines what value is used as a target when calculating this change in tap ratio. **Middle** is the default and means that the target is the average of the regulation minimum and maximum regardless of whether the voltage is high or low. **Max/Min** means that the regulation maximum is used as the target when the regulated voltage is above the maximum, and regulation minimum is used as the target value when the regulated voltage is below the minimum.

### Present Tap Ratio

The tap ratio of the transformer for the current system state.

### Minimum Tap Ratio, Maximum Tap Ratio

Minimum and maximum allowable off-nominal tap ratios for the LTC transformer. Typical values are 0.9 and 1.1.

### Tap Step Size

Transformer off-nominal turns ratio increment. The off-nominal turns ratio is either incremented or decremented from 1.0 in integer multiples of this value. Default value is 0.00625.

### Voltage to Tap Sensitivity

Shows the sensitivity of the voltage magnitude at the regulated bus to a change in the transformer's tap ratio. You can use this field to assess whether or not the transformer can effectively control the regulated bus voltage. In an ideal case, such as when the LTC transformer is being used to control the voltage at a radial load bus, the sensitivity is close to 1.0 (or -1.0 depending upon whether the tapped side of the transformer is on the load side or opposite side of the transformer). However, sometimes the transformer is very ineffective in controlling the voltage. This is indicated by the absolute value of the sensitivity approaching 0. A common example is a generator step-up transformer trying to control its high-side voltage when the generator is off-line. Simulator automatically disables transformer control if the transformer sensitivity is below the value specified on *Power Flow Solution Tab* of the [Simulator Options dialog](#).

### Impedance Correction Table

This field specifies the number of the transformer's corresponding transformer impedance correction table. Transformer impedance correction tables are used to specify how the impedance of the transformer should change with the off-nominal turns ratio. If this number is 0, then no impedance correction table is associated with the transformer, and the impedance of the transformer will thus remain fixed as the tap ratio changes. Valid impedance correction table numbers range from 1 to 63. To assign an existing impedance correction table to the transformer, enter the existing table's number. To view the existing impedance correction tables, click the *Insert/View Impedance Correction Table* button, which brings up the [Transformer Impedance Correction Dialog](#). To define a brand new impedance correction table for the transformer, enter an unused table number and then click *Insert/View Impedance Correction Table* to prescribe the correction table. Note that the association between a transformer and an impedance correction table is not finalized until you select either *OK* or *Save* on the [Line/Transformer Dialog](#).

### View Transformer Correction Table or Insert Transformer Correction Table

Click on this button either to view or to insert transformer correction tables. Clicking on this button displays the [Transformer Impedance Correction Dialog](#). Note that the table must prescribe at least two points in order to be defined.

---

## Transformer Mvar Control Dialog

---

The Transformer Mvar Control dialog is used to view the control parameters associated with load-tap-changing (LTC) transformers that are used to control the Mvar flow through the transformer. To view this display, click on the **Automatic Control Options** button on the [Line/Transformer Information Dialog](#).

When used to control reactive power, the LTC transformer always controls the reactive power flow at the *from* end of the transformer (i.e., the tapped side), with positive flow assumed to be going through the transformer to the *to* bus. Therefore the regulated bus field is not used.

The dialog has the following fields:

### Mvar Flow at *From* Bus

The current Mvar flow as measured at the *from* end of the line. This is the parameter the transformer tries to control.

### Mvar Error

If the Mvar flow at the *from* end violates the limits defined by the *Minimum Mvar Flow* and *Maximum Mvar Flow* fields, the *Mvar Error* field indicates by how much the flow falls outside the control range.

### Regulation Minimum Mvar Flow, Regulation Maximum Mvar Flow

Minimum and maximum allowable reactive power flow as measured at the *from bus*. The transformer attempts to regulate the reactive flow to fall within this range.

### Regulation Target Type

As long as the regulated Mvar flow is inside the regulation minimum and maximum, then the transformer will not change its tap ratio. When the regulated Mvar flow moves outside of this regulation range, then Simulator will calculate a new tap ratio in an attempt to bring the regulated Mvar flow back inside of its range. The **Regulation Target Type** determines what value is used as a target when calculating this change in tap ratio. **Middle** is the default and means that the target is the average of the regulation minimum and maximum regardless of whether the Mvar flow is high or low. **Max/Min** means that the regulation maximum is used as the target when the regulated Mvar flow is above the maximum, and regulation minimum is used as the target value when the regulated Mvar flow is below the minimum.

### Present Tap Ratio

The transformer's present off-nominal turns ratio.

### Minimum Tap Ratio, Maximum Tap Ratio

Minimum and maximum allowable off-nominal tap ratios for the LTC transformer. Typical values are 0.9 and 1.1.

### Tap Step Size

Transformer off-nominal turns ratio increment. The off-nominal turns ratio is either incremented or decremented from 1.0 in integer multiples of this value. Default value is 0.00625.

### Mvar to Tap Sensitivity

The amount of Mvar shift that would be implemented by switching one tap position from the current position. This sensitivity indicates the ability of the transformer to control Mvars.

### Impedance Correction Table

Specifies the number of the transformer's corresponding transformer impedance correction table. Transformer impedance correction tables are used to specify how the impedance of the transformer should change with the off-nominal turns ratio. If this number is 0, no impedance correction table is associated with the transformer, and the impedance of the transformer will thus remain fixed as the tap ratio changes. Valid impedance correction table numbers range from 1 to 63. To assign an existing impedance correction table to the transformer, enter the existing table's number. To view the existing impedance correction tables, click the *Insert/View Impedance Correction Table* button, which brings up the [Transformer Impedance Correction Dialog](#). To define a brand new impedance correction table for the transformer, enter an unused table number and then click *Insert/View Impedance Correction Table* to prescribe the correction table. Note that the association between a transformer and an impedance correction table is not finalized until you select either *OK* or *Save* on the [Line/Transformer Dialog](#).

#### **View Transformer Correction Table or Insert Transformer Correction Table**

Click on this button either to view or to insert transformer correction tables. Clicking on this button displays the [Transformer Impedance Correction Dialog](#). Note that the table must prescribe at least two points in order to be defined.

---

## Transformer Phase Shifting Information

---

The Transformer Phase Shifting Dialog is used to view the control parameters of phase-shifting transformers. To view this display, click on the **Automatic Control Options** button on the [Line/Transformer Information Dialog](#), provided that the *Phase Shift Control* option is chosen from the *Automatic Control* group.

### Regulated Bus Number

Number of the terminal bus of the phase shifter regulated by the phase shifter. When control is active, the phase shifter will automatically change its phase shift to keep the MW flow at this bus at the desired value.

### Current MW Flow

Current MW flow through the transformer measured at the regulated bus terminal.

### MW Error

If the current MW flow falls outside the minimum/maximum MW flow limits, the *MW Error* field indicates by how much the flow violates the regulating range.

### Regulation Minimum MW Flow, Regulation Maximum MW Flow

Minimum and maximum allowable MW flow through the phase shifter.

### Regulation Target Type

As long as the regulated MW flow is inside the regulation minimum and maximum, then the transformer will not change its tap ratio. When the regulated MW flow moves outside of this regulation range, then Simulator will calculate a new tap ratio in an attempt to bring the regulated MW flow back inside of its range. The **Regulation Target Type** determines what value is used as a target when calculating this change in tap ratio. **Middle** is the default and means that the target is the average of the regulation minimum and maximum regardless of whether the MW flow is high or low. **Max/Min** means that the regulation maximum is used as the target when the regulated MW flow is above the maximum, and regulation minimum is used as the target value when the regulated MW flow is below the minimum.

### Present Phase Angle (Degrees)

The phase angle of the transformer for the current solved system state.

### Minimum Phase Angle, Maximum Phase Angle

Minimum and maximum allowable phase shift in degrees.

### Step Size (Degrees)

Phase shift change per step in degrees.

### MW Flow to Phase Sensitivity

The sensitivity of the controlled MW flow to changes in the transformer's phase. This sensitivity indicates the transformer's ability to regulate its MW flow.

### Impedance Correction Table

Specifies the number of the transformer's corresponding transformer impedance correction table. Transformer impedance correction tables are used to specify how the impedance of the transformer should change with the off-nominal turns ratio. If this number is 0, no impedance correction table is associated with the transformer, and the impedance of the transformer will thus remain fixed as the tap ratio changes. Valid impedance correction table numbers range from 1 to 63. To assign an existing impedance correction table to the



transformer, enter the existing table's number. To view the existing impedance correction tables, click the *Insert/View Impedance Correction Table* button, which brings up the [Transformer Impedance Correction Dialog](#). To define a brand new impedance correction table for the transformer, enter an unused table number and then click *Insert/View Impedance Correction Table* to prescribe the correction table. Note that the association between a transformer and an impedance correction table is not finalized until you select either *OK* or *Save* on the [Line/Transformer Dialog](#).

#### **View Transformer Correction Table or Insert Transformer Correction Table**

Click on this button either to view or to insert transformer correction tables. Clicking on this button displays the [Transformer Impedance Correction Dialog](#). Note that the table must prescribe at least two points in order to be defined.

---

## Transformer Impedance Correction Tables Dialog

---

The Transformer Correction Tables Dialog is used to view information about the transformer impedance correction tables. These tables are used on some LTC or phase shifting transformers to model the impedance of the transformer as a function of the off-nominal turns ratio or phase shift. The dialog has the following fields:

### Transformer Impedance Correction Table Number

Number of the impedance correction table, between 1 and 63. Use the spin button immediately to the right of this field to step through the list of defined tables. If you have made changes to a particular table, you must click *Save* before moving to another correction table; otherwise, your changes will be lost.

### Table Name

The name assigned to the table.

### Table Entries

Used to insert/edit/delete the actual entries in the impedance correction table. In the first row, enter either an off-nominal turns ratio for an LTC transformer, or a phase shift in degrees for a phase shifting transformer. The entries in the first row must be entered in strictly ascending form. In the second row, enter the scale factor to apply to the transformer impedance. The transformer's nominal impedance is multiplied by the scale factor to obtain the actual value. Note that at least two columns must be used.

Right-click on the table to invoke its local menu, which allows you to delete and to insert columns. To insert a new column, click on the column before which you want to insert a new column and select **Insert New Point** from the local menu. To delete a column, position the cursor on the column you want to delete and select **Delete Point**.

### Table is Used by the Following Transformers

Lists all the transformers in the case that use this impedance correction table. A single table may be used by any number of transformers. To associate a table with a transformer, use the [Transformer AVR Dialog](#) for LTC transformers or the [Transformer Phase Shifting Dialog](#) for phase shifters.

---

**Switched Shunt Information (Run Mode)**


---

The Switched Shunt Information Dialog box can be displayed by placing the cursor on its symbol and right-clicking. This is very similar to its [Edit Mode](#) counterpart. The dialog has the following fields:

**Bus Number**

Unique number between 1 and 2,147,483,647 (equals  $2^{31}$  minus 1) used to identify the bus to which the switched shunt is attached. This drop down list identifies the buses in the case with switched shunts that also have [valid display filters](#). Use the spin button to step through the list of shunts in the case. Note that only one switched shunt is allowed at each bus.

**Find By Number**

To find a switched shunt by its bus number, enter the number into the Bus Number field. Then click the **Find By Number** button.

**Bus Name**

Unique alphabetic identifier for the bus to which the switched shunt is attached, consisting of up to eight characters. This dropdown box lists the names of all the switched shunt buses in the case with [valid display filters](#).

**Find By Name**

To find a switched shunt by its name, enter the bus name into the Bus Name field (case insensitive). Then click the **Find By Name** button.

**Shunt ID**

Since multiple switched shunts are allowed on a single bus, each shunt is identified by a unique ShuntID.

**Find...**

If you do not know the exact switched shunt bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

**Status**

Status of the switched shunt, either Closed (connected to terminal bus) or Open (not connected). On the oneline, the switched shunt can be opened by placing the cursor on the (red) circuit breaker box and clicking, and it can be closed by placing the cursor on the (green) box and again clicking. You can also use this status field to change the switched shunt's status. Note that the switched shunt is only available for automatic control when its status is closed.

**Labels**

Clicking on this button will open the [Subscribed Aliases dialog](#) listing all the labels or aliases assigned for the selected load.

**Parameters****Nominal Mvar**

Amount of reactive power that would be supplied by the switched shunt if its terminal voltage were one per unit (capacitive is positive).

**Actual Mvar**

Actual reactive power in Mvar being injected into the system by the switched shunt (capacitive is positive). The Actual Mvar field is equal to the Nominal Mvar field multiplied by the square of the terminal buses per unit voltage.

### Nominal MW

This field is only visible when a switched shunt object has been read from a file as a Bus Shunt. In that case, it is possible for the bus shunt to have both a MW and MVAR component. The MW component will be displayed here. In general, switched shunts of other control types do not have MW components, and this field will not be displayed. If you change a switched shunt read as a bus shunt to another form of control, the MW component will remain, but has no controllability.

### Actual MW

This field is only visible when a switched shunt object has been read from a file as a Bus Shunt. The value displayed is the actual real power in MW being injected into the system by the shunt. The Actual MW field is equal to the Nominal MVAR field multiplied by the square of the terminal buses per unit voltage.

### Control Mode

Determines whether the switched shunt has a fixed value, or whether the amount of reactive power supplied by the device changes either in discrete steps or continuously in order to maintain its terminal voltage within the voltage range specified in the Voltage Regulation fields. This field can be changed (except in Viewer). However, for a switched shunt to be used for automatic control, three fields must be set correctly:

1. The *Control Mode* field must be set to either *Discrete* or *Continuous*
2. The corresponding area's *Auto Shunts* property must be true, and
3. The case-wide *Disable Switched Shunt Control* option, which can be set on the *Power Flow Solution Tab* of the [Simulator Options Dialog](#), must not be checked.

**Note: automatic control of switched shunts is disabled if the voltage regulation high value is not greater than the low value; they should not be equal unless in the continuous mode.**

Note the additional control mode called Bus Shunt (Fixed). This is analogous to the shunt MW and MVAR values that can also be stored at the bus level. The difference is that bus shunts stored directly with the bus cannot be turned on and off in the load flow; rather they are always included in the load flow solution. Bus shunts that are represented as switched shunt objects on Bus Shunt control, however, are mathematically exactly the same and can be turned on or off. The reason for the differentiation of the Bus Shunt versus normal Fixed control is that the Bus Shunt control type is intended to identify the difference between a bus shunt and a transformer that MAY have controllability, but is currently turned off control by being set to a Fixed value.

### Control Regulation Settings

#### Voltage Regulation

When the switched shunt is on automatic control, its reactive power is changed in discrete steps or continuously to keep the voltage at the regulated bus within the per unit voltage range defined by **High Value** and the **Low Value**.

In the case of discrete control, the amount of reactive power supplied by this device changes in discrete amounts, thus the High Value should be greater than the low value. The necessary voltage range depends upon the size of the switched shunt blocks. In addition to a voltage range for discrete control, a specific **Target Voltage** can be specified as well. The target voltage will try to be met, either approximately under discrete control, or exactly under continuous control (either true continuous or discrete

with a continuous shunt correction element.) The number of the regulated bus is shown in the **Reg. Bus #** field.

### Generator Mvar Regulation

This option allows switched shunts to control generator Mvar output to better enable full var range inside the [inner power flow loop](#). During the first inner power flow loop, generator var limit checking is disabled for generators whose Mvar output is controlled by switched shunts. This will enable the use of the full reactive range of the switched shunts before generators hit their reactive limits. This corrects a problem in which the power flow fails to solve because generators are at their Mvar limits even though there is still reactive support available from switched shunts.

### Switched Shunt Blocks

The amount of shunt reactive power (susceptance) is specified in the Switched Shunt Block field. The columns in this field correspond to different blocks of reactive power. The first row indicates the number of steps in each block, and the second row gives the amount of nominal Mvars per step (assuming 1.0 per unit voltage). You may model both capacitors and reactors. The reactors should be specified first, in the order in which they are switched in, followed by the capacitors, again in the order they are switched in. The sign convention is such that capacitors are positive and reactors negative. Shunt blocks are switched in order from left to right.

## Control Options

### Single Largest Step

This option only applies when a switched shunt is set on discrete control. If checked the switched shunt will switch in EITHER all of the available reactor blocks OR all of the capacitor blocks at once when the voltage falls outside the given range. Whether the reactor or capacitor blocks switch is determined by which limit is violated. A switched shunt with this option checked will only switch ONCE during a load flow solution, and then remains fixed at the new output for the remainder of the same solution calculation.

### Allow switching in the inner power flow loop

This option is available for individual discrete shunts only. If this option is set, discrete shunts are treated as continuous in the [inner power flow loop](#). This means that they are treated as PV buses in the inner power flow loop. After the first inner power flow loop, the shunt nominal Mvar setting is rounded to positive infinity to the next discrete step. If any shunts exist that are allowed to switch in the inner power flow loop, the inner power flow loop is repeated again with the shunts being treated as discrete in the subsequent inner power flow loop. Shunts that are allowed to switch in the inner power flow loop will only switch if the global option to "Disable Treating Continuous SSs as PV Buses" is not checked.

### Use Continuous Element

If this option is checked, then Simulator will use a continuous element to fine-tune a discrete controlled switched shunt by injecting or absorbing additional MVARs to try and obtain the target voltage of the controlled bus.

### Minimum and Maximum Susceptance

The minimum and maximum susceptance range for the continuous correction element.

### Use High Target Voltage

Check this box to use the target voltage specified in the **High Target Value** edit box when the regulated point goes above the High limit. This will give a different target value if the voltage goes out of range on the high end than the low end. If the voltage goes out of

range on the low end, the original target value on the Parameters page will be used. If this box is unchecked, only the target value on the parameters page will be used, whether the violation is high or low.

### **Fault Information**

Typically switched shunts are treated as open circuits in the zero sequence data for fault analysis. However, it is possible to define zero sequence admittance blocks to be used. The blocks work similarly to the load flow Switched Shunt Blocks discussed above. Usually there will be the same number of blocks in the zero sequence data as in the load flow data. Simulator will determine how many blocks were switched in for the power flow solution, and then use the zero sequence block data to calculate the zero sequence admittance for the same number of steps and blocks.

### **Owner, Area, Zone, Sub**

This tab is used to display or change the generator's owner information, area information, zone, and substation information

#### **Area Number, Area Name**

The area number and name to which the generator belongs. Note that you can change the area of the generator to be different than the area of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the generator within a different area than that of the bus to which it is electrically connected.

#### **Zone Number, Zone Name**

The zone number and name to which the generator belongs. Note that you can change the zone of the generator to be different than the zone of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the generator within a different zone than that of the bus to which it is electrically connected.

#### **Owner Number, Owner Name**

The owner number and name to which the switched shunt belongs. Note that you can change the owner of the switched shunt to be different than the owner of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the switched shunt within a different owner than that of the bus to which it is electrically connected.

#### **Substation Number, Substation Name**

The substation number and name to which the switched shunt belongs. Note that you can change the substation of the switched shunt to be different than the substation of the terminal bus. If you do so, you will be prompted to confirm that you wish to place the switched shunt within a different substation than that of the bus to which it is electrically connected.

### **Custom**

This page of the dialog can be used to enter notes about the switched shunt. Any information entered in the memo box will be stored with the case when the case is saved to a PWB file. Custom fields can also be viewed and added from this page as well.

---

## Zone Information (Run Mode)

---

This dialog is used in the Run Mode to view and modify information associated with a zone record. It displays different information from the [Edit Mode version of the zone dialog](#). To view this dialog, open the [Zone Records Display](#) by clicking on **Aggregations > Zones** in the [Model Explorer](#). Right-click on the zone of interest and choose **Show Dialog**. The Run Mode Zone Dialog has the following fields:

### Zone Number, Zone Name

Number and name of the associated zone. Use either the combo box or the spin arrows to view the different zones.

### Find...

If you do not know the exact zone number you are looking for, you can click this button to open the [advanced search engine](#).

### Labels

To assign alternative identifying labels to the zone, click the Labels button, which will open the [Label Manager Dialog](#) listing all the labels or aliases assigned for the selected zone.

The rest of the Run Mode Zone Dialog is divided into multiple pages:

## Information

### Load and Generation

Real and reactive load, generation, shunts, losses, and interchange for the zone.

### Generation AGC Range

These two fields show the total available MW reserve for generators in the zone that are on AGC and have nonzero participation factors. In other words, these fields show the total MW by which the generation in the zone can be increased or decreased using only generation that is presently on-line. The generator status, AGC status, and participation factor can be changed on the [Generator Records Display](#).

## Tie Lines

### Zone Tie Lines

The Zone Tie Line Table identifies the flows on all of the zone's ties to other zones.

## OPF

### Average LMP for Zone

The computed average locational marginal price of all buses contained in the zone.

### LMP Standard Deviation

The standard deviation of the locational marginal price for all buses contained in the zone.

### Minimum LMP

The minimum locational marginal price of all the buses in the zone.

### **Maximum LMP**

The maximum locational marginal price of all the buses in the zone.

### **Zone Buses, Zone Gens, Zone Loads, Zone Switched Shunts**

These pages show the case information display with the buses, generators, loads and switched shunts assigned to the zone.

### **Custom**

The Custom Page of the Zone Information dialog displays custom numbers or strings defined with the viewed zone. The [Memo](#) box is simply a location to log information about the zone. To log information about the zone, simply start typing your information or comments about the zone in the memo box.



---

## Labels

---

### Using Labels for Identification

Most data objects (such as buses, generators, loads, switched shunts, transmission lines, areas, zones, and interfaces) may have an alternative names assigned to them. These alternative names are called labels. Labels allow you to refer to equipment in the model in a way that may be unique to your organization. Labels may thus help clarify which elements are described by a particular set of data, especially when the short names employed by the power system model prove cryptic. Furthermore, since labels are likely to change less frequently than bus numbers, and since a label must, by definition, identify only one power system component, they may function as an immutable key for importing data from [auxiliary files](#) into different cases, even when bus numbering schemes change between the cases. Labels must be unique for devices of the same type, but the same label can be used for a device of a different type.

Information dialogs corresponding to buses, generators, loads, switched shunts, transmission lines, areas, zones, and interfaces feature a button called **Labels**. If you press this button, the device's Label Manager Dialog will appear. The [Label Manager Dialog](#) lists the labels associated with the device and allows adding, deleting, and modifying labels. The **Labels (All)** field (variablename = LabelsAll) in a [Case Information Display](#) lists all of the labels that have been associated with a device. The device's primary label is the one that is listed first in this field. This field lists all labels assigned to a device as a comma-delimited string. Even though multiple labels may be assigned to a device, any label can be used to import data from [auxiliary data files](#). The Label Manager Dialog can be used to assign labels or the string of labels can be entered directly in the **Labels (All)** field.

Labels can be used to map data from an auxiliary data file to a power system device. Recall that [auxiliary data files](#) require you to include a device's key fields in each data record so that data may be mapped to the device. Labels provide an alternative key. Instead of supplying the bus number to identify a bus, for example, you can supply one of the bus's labels. The label will enable Simulator to associate the data with the device associated with that label. This mechanism performs most efficiently when the primary label is used, but other labels will also provide the mapping mechanism. The **Label (for use in input from AUX or Paste)** field (variablename = Label) is used for importing data using labels and is blank when viewing in a case information display. Keep in mind that all devices read via an auxiliary file using the label field should have a non-blank label. Otherwise, information for that device will not be read. Even if the primary or secondary key fields are provided with the device, as long as the label field is present, that is the only field that will be used to identify the device. New devices cannot be created by simply identifying them by label. Either the primary or secondary key fields must be present to create a new device and the label field should not be present.

Again, it is important to remember this: a single power system device may have multiple labels, but each label may be associated with only one device of a particular type. This is the key to enabling data to be imported from an auxiliary file using labels.

### Saving Auxiliary Files Using Labels

All devices that can be identified by labels will have the **Labels (All)** and **Label (for use in input from AUX or Paste)** fields available in their case information displays. In order to save auxiliary files that identify devices by label, the two label fields should be added to the case information display prior to saving the data in an auxiliary file. Because the **Label (for use in input from AUX or Paste)** field will be blank when saved in the auxiliary file, this field must be populated with one of the labels in the **Labels (All)** field before loading the auxiliary file back in. Keep in mind that devices with blank labels cannot be identified when

loading in an auxiliary file, so avoid saving auxiliary files by label if all devices do not have labels.

Many devices require [SUBDATA](#) sections. These sections have custom formats specific to the type of information that they contain. When saving auxiliary files with devices that require SUBDATA sections, the user can choose to use primary or secondary key fields or labels to identify devices in the SUBDATA sections. The user will either be prompted when saving the devices, or there is an option to change the key field to use when saving subdata sections on the PowerWorld Simulator Options dialog under the [Case Information Displays](#) category. When choosing to use labels, if a device has a label, it will be used. If it is a device that can be identified by buses and bus labels exist, bus labels will be used. Finally, if the device does not have a label and the buses do not have labels, the primary key for the device will be used for identification.

Devices that have SUBDATA sections that contain other devices that can be identified by labels include: contingencies, interfaces, injection groups, post power flow solution actions, and owners.

The setting to choose which identifier to use for the SUBDATA sections does not just apply to SUBDATA sections. Often when saving groups of options, this setting will apply to everything being saved with those options and not just the SUBDATA sections. This includes contingency options, ATC options, limit monitoring settings, and PVQV options. In these cases, there will be a prompt asking the user to decide which identifier to use in the auxiliary file.

### Loading Auxiliary Files SUBDATA Sections Using Labels

The various SUBDATA sections that represent references to other objects can also be read using labels. Examples include contingencies, interfaces, injection groups, post power flow solution actions, and owners. When reading a SUBDATA section such as this, PowerWorld makes no assumption ahead of time about what identification was used to write this SUBDATA section. Instead, an order of precedence for the identification is as follows

	Identification	Explanation	Example
1	Key Fields	assumes that the strings represent Key Fields	BRANCH 8 9 1
2	Secondary Key Fields	assumes that the strings represent Secondary Key Fields	BRANCH Eight_138 Nine_230 1
3	Labels for component objects	the key/secondary key fields for some objects consist of references to other objects. An example of this is the BRANCH object that is described by the From Bus, To Bus, and Circuit ID. This assumes that labels of the component objects are used.	BRANCH Label8 Label9 1
4	Labels	Assumes that the string represents one of the Labels of the object	BRANCH LabelForBranch

### Special Use of Labels

There are a few special cases where objects have fields that identify other devices. These devices can be identified by label but not in the conventional means because the label field applies to the object that contains the device and a SUBDATA section is not necessary. These special cases include: (Note all fields given below are by variable name because the use of labels is most relevant with auxiliary files.)

### ATC Scenarios

[ATC Scenario](#) change records usually contain primary key fields to identify the devices that should be adjusted during the scenario. If using labels, these primary key fields will be replaced with a single **Label** field. The use of this field is different because the Label field refers to the device in the change record and not to the change record itself. When labels are used with ATC scenarios, device labels only can be used. Bus labels cannot be used to identify devices for which no label exists but a bus label does.

ATC Scenarios are saved in an auxiliary file if choosing to Save Settings on the [ATC Dialog](#).

### ATC Extra Monitors

[ATC Extra Monitors](#) identify either branches or interfaces to monitor during the ATC analysis. These devices are identified in the **WhoAmI** field of ATC Extra Monitor records. Usually, the **WhoAmI** field is a special format that contains key field tags. Optionally, this field can use the label of the device for the extra monitor. If the device label is not available, the standard format will be used. There is no option to use bus labels if they exist and the device labels do not.

ATC Extra Monitors area saved in an auxiliary file if choosing to Save Settings on the [ATC Dialog](#).

### Model Conditions

Devices in [Model Conditions](#) are usually identified by the **WhoAmI** field which is in a special format that contains key field tags. Optionally, this field can use the label of the device. If the device label does not exist, the standard format will be used. There is no option to use bus labels if they exist and the device labels do not.

### Model Expressions

[Model Expressions](#) contain Model Fields. Model Fields are identified by the **WhoAmI** fields in the Model Expressions. Usually, the WhoAmI fields are in a special format that contains key field tags. Optionally, these fields can use the label of the device associated with the Model Field. If the device does not exist, the standard format will be used. There is no option to use bus labels if they exist and the device labels do not.

### Bus Load Throw Over Records

[Bus Load Throw Over Records](#) are used with contingency analysis. These records have an option to identify the bus to which the load will be transferred by either number or name\_kV combination. If choosing to identify objects by label, the **BusName\_NomVolt:1** field will contain the label of the bus instead of the name\_kV combination.

Bus Load Throw Over Records will be saved in an auxiliary file if choosing to [Save](#) settings on the Contingency Analysis dialog.

### Injection Group Participation Points

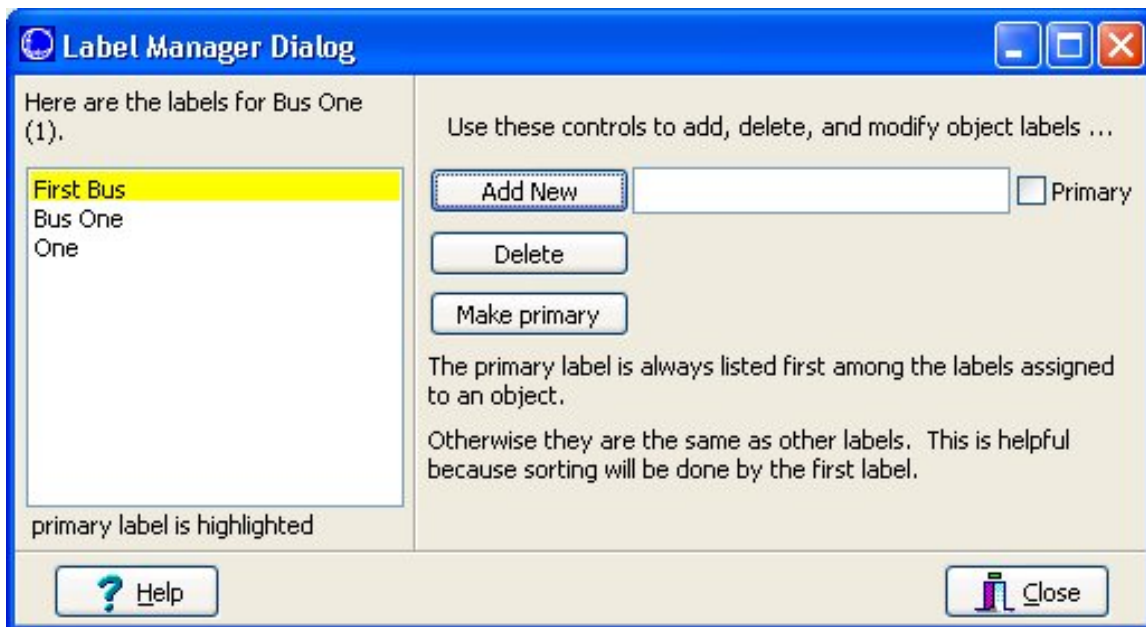
All participation points and the injection groups to which they belong can be listed on the [Injection Group Display](#). Load, generator, and shunt devices that can be assigned to a participation point must be identified by bus and ID. The bus can be identified by either the number or name. When identifying by name, the **BusName\_NomVolt** field is used to provide the name\_kV combination for the bus. If choosing to identify devices by label, this field instead will contain the label of the device. If the device does not have a label but the bus does, the bus label will be used instead in conjunction with the ID of the device. Even if the device does contain a label, the ID field must be included in any auxiliary file that is going to be loaded because it is a key field. Injection groups can be included in other injection groups. Injection groups can be identified by label, even though this is not a normal thing to do. If any injection groups have labels and these injection groups are included in other injection groups, their labels will also appear in the

BusName\_NomVolt field. If they do not have labels, they will be identified by the injection group name that appears in the PPntID field.

## Label Manager Dialog

The Label Manager Dialog allows the creation and modification of [labels](#) to identify power system devices. Labels can be used for most data objects (such as buses, generators, loads, switched shunts, transmission lines, areas, zones, and interfaces) as alternate names for these devices. See the [Labels](#) topic for more information on how labels can be used.

The Label Manager Dialog can be accessed by clicking the Labels button on any of the information displays for objects that allow labels.



### List of Labels

Multiple labels can be assigned to a single object. The list on the left of the dialog shows all of the labels that have been assigned with the primary label listed first and highlighted in yellow.

### Add New

Click this button to add a new label after entering the new label in the box to the right of the button. If this new label should be the primary label, check the **Primary** box. Labels must be unique for each type of object. That means that there cannot be two buses labeled *One*. However, there could be objects of different types with the same label. There could be a bus and a generator both labeled *One*. If attempting to label an object with a label that has already been used by the same object type, the label will be assigned to the current object and removed from the object to which it was previously assigned.

### Delete

Click this button to delete a label after first selecting the label in the list of labels.

### Make Primary

Click this button to make a label the primary label after first selecting the label in the list of labels. The primary label will be moved to the top of the list. The primary label will always appear at the top of the list and will be used for sorting when sorting by label.

---

## Area Information

---

The Area Information Dialog shows information about each area in the system. It displays an area's load, generation and losses; the area's scheduled interchange with other areas; options for controlling the area's generators, transformers, and shunts; the flows on its tie lines; and its operating cost information. You may view this dialog by doing any of the following:

- Select Areas from the [Model Explorer](#). This displays the [Area Display](#). Select the desired area and click the Show Dialog option on the [case information toolbar](#), or right-click on the record corresponding to the desired area to bring up the display's local menu and choose Show Dialog.
- Right-click in an empty portion of the online near a bus in the area of interest to display the [online's local menu](#). Select Area Information Dialog.
- While in Run Mode, right-click on an [Area Display Object](#) that represents an area and choose Information Dialog. Note: Doing this while in Edit Mode will show the [Area Display Options](#) dialog instead of the Area Information Dialog.

The Area Information Dialog contains the following information:

### Number

A drop-down box that specifies the area number. Select an area from the drop-down box, or use the spin button to cycle through the list.

### Find By Number

To find an area by its number, type the number in the **Number** field and click this button.

### Name

A drop-down box that specifies the area's alphabetic identifier, which may be any length in Simulator. When saving area names to other load flow formats, the names are truncated at the maximum character length supported by that format.

### Find By Name

To find an area by its name, type the name in the **Name** field and click this button.

### Find...

If you do not know the exact area number or name you are looking for, you can click this button to open the [advanced search engine](#).

### Super Area

To associate an area with a particular super area, either select an existing super area from the drop-down box, or type the name of a new super area.

### Area Control Options

Select to change the area's method of [Automatic Generation Control \(AGC.\)](#)

### Labels

To assign alternative identifying labels to the area, click the Labels button.

The rest of the Area Information Dialog is divided into nine pages of controls:

- [Info/Interchange](#)
- [Options](#)
- [Area MW Control Options](#)
- [OPF](#)
- [Tie Lines](#)
- [Buses](#)
- [Gens](#)

## Properties of Simulator Objects

- [Loads](#)
- [Custom](#)



---

**Area Information: Buses**

---

The Area Buses grid shows a list of all buses that are in the area. The grid displays the bus number, bus name and other bus information. The Area Buses display is a [Case Information Display](#) and has options available to these types of displays.

---

**Area Information: Gens**

---

The Area Gens grid shows a list of all generators that are in the area. The grid displays the bus number, bus name, generator ID and other generator information. The Area Gens display is a [Case Information Display](#) and has options available to these types of displays.

---

**Area Information: Loads**

---

The Area Loads grid shows a list of all loads that are in the area. The grid displays the bus number, bus name, load ID and other load information. The Area Loads display is a [Case Information Display](#) and has options available to these types of displays.

---

## Area Information: Area MW Control Options

---

### Set Participation Factors...

Press this button to open the [Generator Participation Factors Dialog](#), which gives you control over how the participation factor for each generating unit is defined. If you decide not to prescribe participation factors using this button, the existing participation factors defined with each generator will be used. This button will be enabled only if the area has been set to Participation Factor area control.

### Area Slack Bus Number

Identifies which bus to model as the area's slack bus.

### Injection Group Area Slack

An additional area generation control option that can be utilized is the ability for Simulator to use a defined [injection group](#) as the sink for Area Slack control. When an injection group is selected, and the area is set on Area Slack control, any needed injection changes in the area will be accounted for by the devices in the injection group.

Some typical injection group options are also available here for handling the injection group dispatching during the Injection Group Area Slack Control. These options include:

#### **Allow only AGC gen/load to vary:**

Only generators and loads in the injection group that are designated as available for AGC control will be allowed to participate in injection changes.

#### **Enforce generator MW limits (ignore case and area options):**

Generators in the injection group can be designated to enforce their MW limits independent of the global case and area options for generator MW limit enforcement.

#### **Do not allow negative loads:**

If loads are decreasing demand as part of an injection group area dispatch, you can specify whether or not those loads should be allowed to go below 0 demand.

#### **How should reactive power load change as real power load is ramped?**

You can choose to keep the ratio of real and reactive power constant for each load that is included in the injection group, or you can specify a constant power factor that the MVAR value will be determined from when the MW value is changed.

---

## Area Information: Info/Interchange

---

The Info/Interchange Tab serves as an accounting sheet for flows into and out of an area. It houses the following controls:

### Load, Generation, Shunts, Losses, Interchange

These read-only fields express the total real and reactive load, generation, shunt compensation, losses, and interchange for the area.

### Interchange

Interchange in MW between the area and all other areas (exporting power is positive). To prescribe an interchange with a specific area, simply enter the amount of power in MW to export to that area in the MW Export column. To specify an import, enter a negative value. If the recipient of the power is unknown, you may enter the net total in the **Unspecified MW Interchange** field found above the list of transactions. The net value of the MW imports and exports is displayed in **Transaction MW**.

Check the box **Only Show Areas with Nonzero Interchange** to display only those areas whose interchange is different from zero.

By default, one interchange record is initially defined between each area. The total interchange between two areas can be managed using this single record. However, it is possible to define multiple interchange records between the same areas. To do so, you can right-click in the interchange table and select Insert from the local menu. This will open the [Transaction Dialog](#), which will allow you to insert a new transaction. Transactions between the same two areas must have unique transaction IDs. Once you have multiple transactions defined between the same two areas, you can choose which transactions are enabled using the Enabled property in the Base Interchange table. This field is a toggleable field, meaning you can double-click in the field to toggle its value.

The total interchange defined for an area will be displayed in the summary field labeled **Interchange** in the area summary totals on the left side of the page.

### AGC Tolerance

The MW tolerance is used in enforcing area interchange. When the absolute value of the ACE is less than this value, Simulator considers the area interchange constraint to be satisfied.

### ACE (Area Control Error)

Current area control error (ACE) for the area in MW. Note that, when the constant frequency model is used,  $ACE = \text{area generation} - \text{area load} - \text{area losses} - \text{scheduled area interchange} - \text{area shunt MW}$ .

### Area Has Multiple Islands

This is an informational field that cannot be changed. If checked, then Simulator has detected that the area has devices in separate [electrical islands](#) in the system. This is important because it is sometimes impossible for an area using some form of generation control ([AGC](#)) to solve for the generation dispatch in the entire area ( $\text{Area Generation} = \text{Area Load} + \text{Area Losses} + \text{Area Interchange} + \text{Area Shunt MW}$ ) when the area is not entirely within the same electrical island. If Simulator is unable to solve for areas that span multiple islands, the AGC status of the area will automatically be set to "Off AGC" and a warning message will be written to the Simulator [message log](#).

---

## Area Information: Options

---

### Report Limit Violations

If checked, limit violations for this area are reported. Limits violations are reported in the [Limit Violations List](#).

### Generation AGC Values

These fields indicate the total available AGC range for all the generators in the area **that are specified as being on AGC and have nonzero participation factors**. That is, these fields show the total amount the generation in the area that can be increased or decreased using the presently on-line generation. This may be referred to as the *spinning reserve*. The generator status, AGC status, and participation factor can be changed on the [Generator Records Display](#).

### Load MW Multiplier Value

This value is used for scaling the area MW load. The base load remains unchanged.

### Load MVAR Multiplier

This value is used for scaling the area MVAR load. The base load remains unchanged.

### Automatic Control Options

The Automatic Control Options section provides a convenient mechanism to enable or disable automatic control of switched shunts and transformers in the area.

This section also provides a check box for setting if the generators in the area should enforce their Generator MW Limits or not. Typically this should be checked. Note that even if unchecked, the [OPF](#) routine will ignore this setting and will always enforce generator MW limits during an OPF solution.

### Include Loss Penalty Factors in ED

If this box is unchecked, then the economic dispatch for the area is calculated assuming that the area is lossless. Otherwise, the solution will incorporate losses when computing the economic dispatch. The penalty factors gauge the sensitivity of the area's losses to changing injection at specific generators. The option to calculate loss penalty factors is relevant only when the area operates according to *Economic Dispatch Control*. Usually, if the system's cost curves are relatively flat, the inclusion of losses in the solution will not have much of an effect on the dispatch.

### Economic Dispatch Lambda

The lambda value calculated during the economic dispatch computation. This field is valid only when the area is on economic dispatch control.

---

**Area Information: Tie Lines**

---

The Tie Lines display shows the flow on all of the tie lines for the selected area. This display is a [case information display](#) and contains many of the options available to these types of displays.

The default fields that are shown in the grid identify each line and show the flow on each line.

**Near Area Name, Near Area Number**

These fields identify the area name and number of the tie line terminal located in the selected area.

**Near Number, Near Name**

These fields identify the bus number and name of the tie line terminal located in the selected area.

**Far Area Name, Far Area Number**

These fields identify the area name and number of the tie line terminal located in the other area.

**Far Number, Far Name**

These fields identify the bus number and name of the tie line terminal located in the other area.

**Ckt**

This is the circuit identifier for the tie line.

**Meter MW, Meter Mvar**

These are the real and reactive power **flowing on the tie line from the selected area to the other area**. (Real and reactive power flowing on line from the Near bus to the Far bus.)

**Status**

This field gives the status of the tie line as *Open* or *Closed*.

**MW Loss, Mvar Loss**

These fields show the real and reactive power losses on the tie line.

---

**Area Information: OPF**

---

The OPF Tab contains information regarding the [Optimal Power Flow \(OPF\)](#) solution data for an area. It houses the following information:

**Average LMP for Area**

The computed average locational marginal price of all buses contained in the area.

**LMP Standard Deviation**

The standard deviation of the locational marginal price for all buses contained in the area.

**Min/Max LMP**

The minimum and maximum locational marginal price of all the buses in the area.

**Total Generator Production Cost (Scaled)**

The scaled cost includes the Cost Shift and Cost Multiplier. These two values can be defined for each generator, and allow the user a way to assess changes to the LMP results when a generator's cost or "bid" is modified, without actually changing the original generator cost or bid curve. The scaled cost function for each generator is equal to:

$(\text{original cost function} + \text{cost shift}) * \text{cost multiplier}$

**Total Generator Unscaled Production Cost**

The total unscaled generator production cost, based on the original generator cost or bid curves.

**Total Generator LMP Profit**

The profit of the generators in the area based on the Locational Marginal Prices (LMPs) determined by the OPF solution. The profit is determined as:

$\text{LMP Price} * \text{MW Output} - \text{Unscaled Cost Function}$

**Cost of Energy, Loss, and Congestion Reference**

Specify a reference for determining the cost of energy, loss and congestion. The choices are Existing loss sensitivities directly, Area's Bus' Loads, [Injection Group](#), or a specific bus.



---

**Area Information: Custom**

---

The Custom page of the [Area Information](#) dialog contains two sections: custom fields and memo.

The custom fields section allows access to setting and changing the values for custom fields that have been defined for areas. Defining custom fields is detailed in [Custom Field Descriptions](#).

The [Memo section](#) of the Area Information dialog is simply a location to log information about the area. To log information about the area, simply switch to the Custom page on the dialog, and start typing your information or comments about the area in the memo box.

---

## Area Field Options

---

Area field objects are used to show different values associated with areas and the system. This dialog is used to view and modify the parameters associated with these fields.

### Area Number

Area number associated with the field. When you insert fields graphically, this field is automatically set to the area number associated with the closest bus on the oneline. With most types of area fields, an *Area Number* of 0 is valid and defines the field as showing values for the entire system.

### Find...

If you do not know the exact bus number or name you are looking for, you can click this button to open the [advanced search engine](#).

### Total Digits in Fields

Total number of digits to show in the field.

### Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

### Rotation Angle in Degrees

The angle at which the text is to appear on the oneline diagram.

### Other Area Number

Some of the fields, such as **MW Flow to Other Area**, require that a second area be specified. If applicable, enter the second (other) area here.

### Other Area Transaction ID

Since it is now possible to have more than one base transaction defined between the same two areas, base transactions must now also have a unique ID to distinguish between transactions among the same two areas. If you are displaying a field that pertains to the display of the scheduled flow between areas, the ID of the transaction in question will also need to be entered here.

### Delta MW per Mouse Click

This value is used only with the **Sched Flow to Other Area** field type. When there is a nonzero entry in this field, and the field type is **Sched Flow to Other Area**, a spin button is shown to the right of the area field. When the up spin button is clicked, the flow to the other area is increased by this number of MW; when the down button is clicked, the scheduled flow is decreased by this amount.

### Field Prefix

A prefix that can be specified and displayed with the selected value.

### Field Value

Shows the current output for the area field. Whenever you change the **Type of Field** selection, this field is updated.

For the **Sched Flow to Other Area** field type only, you can specify a new value in MW. Exports are assumed to be positive.

### Anchored

If checked, the area field will be anchored to a corresponding [Area Object](#). If the area object is moved on the diagram, the text field will move with it.

### Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

### Type of Field

Used to determine the type of bus field to show. The following choices are available:

**Name** Area name (eight characters maximum). The *Area Number* field must correspond to a valid area.

**Number** Area number (1 - 999). The *Area Number* field must correspond to a valid area.

**MW Load, Mvar Load** If the area number is nonzero, then these fields show Total MW or Mvar load for the area. If the area number is zero, these fields show the total load in the entire system.

**MW Generation, Mvar Generation** If the *Area Number* is nonzero, then these fields show Total MW or Mvar generation for the area. If the *Area Number* is zero, these fields show the total generation in the entire system.

**MW Losses, Mvar Losses** If the *Area Number* is nonzero, then these fields show Total MW or Mvar losses for the area. If the *Area Number* is zero, these fields show the total losses in the entire system.

**ACE (MW)** Area Control Error in MW for the area. The *Area Number* field must correspond to a valid area.

**Hourly Cost (\$/hr)** If the *Area Number* is nonzero, then this field shows the hourly cost for the area. If the *Area Number* is zero, these fields show the hourly cost for the entire system.

**Total Cost (\$)** If the *Area Number* is nonzero, this field shows the total cost incurred by the area since the beginning of the simulation. If the *Area Number* is zero, this field shows the total cost incurred throughout the system since the beginning of the simulation.

**MW Flow to Other Area,**

**Mvar Flow to Other Area** Total MW or Mvar flow from the area specified in the *Area Number* field to the area specified in the *Other Area Number* field. The *Area Number* field must correspond to a valid area. If the *Other Area Number* field is zero, this field shows the area's total MW or Mvar exports.

**Sched. Flow to Other Area** Scheduled MW transaction from the area specified in the *Area Number* field to the area specified in the *Other Area Number* field, and with the Transaction ID given in the *Other Area Transaction ID* field. The *Area Number* field must correspond to a valid area. If the *Other Area Number* field is zero, this field shows the area's total scheduled MW transactions. If the *Delta per Mouse Click* value is nonzero, you can use the spin arrows on the oneline to change this value. Also, you may directly enter a new value in the *Field Value* field.

**Load Schedule Multiplier (MW Only)** Indicates the current value of the MW multiplier applied to the area's loads.

**AGC Status** Displays the AGC status of the area.

**Select a Field** Choose from all possible information fields that can be displayed for areas. Select **OK** to save changes and close the dialog or *Cancel* to close dialog without saving your changes.

---

## Super Area Information Dialog

---

The Super Area Information Dialog displays information pertaining to [Super Area Display](#). It summarizes the super area's real and reactive load and generation, scheduled transactions, and constituent areas. It also allows you to manage the list of areas to include in the super area and to designate how its generation should be controlled. To display the Super Area Information Dialog, right-click on the [Super Area Records Display](#) and select **Show Dialog** from the resulting local menu.

The Super Area Information Dialog has the following fields:

### Area in Super Area

#### Name

The list of all super areas that have been defined in the case are listed in this dropdown box. Select one of the super areas to display its information, or type a new name in the box and click **Add New** to define a new super area.

#### Rename

Click this button to change the name of the super area specified in the Name field.

#### Save

Saves the super area of the name specified in the list of super areas.

#### Delete

Click this button to delete the super area specified in the Name field.

### Area in Super Area

#### Areas in Super Area

Lists the areas contained in the super area. Right-click on a row and select **Remove** to remove the area from the super area.

#### Super Area Control Options

Specify the type of generation control to employ for the super area. The super area may be removed from area control or employ Participation Factor or Economic Dispatch control. See [Area Control](#) for details on these types of generation control.

An additional type of control available to Super Areas is to **Use Area Participation Factors**. If the super area is set to participation factor control, and this option is NOT checked, then the generators within the super area respond by redispatching to meet the entire super area generation change based on their own participation factors.

However, if this option IS checked, then an additional level of participation control complexity is added for the super area. First, the total super area generation change is divided across each area forming the super area, according to the participation factors of the AREAS, as set in the grid on this dialog. Once the total generation change has been determined for each area within the super area, then the generators within each individual area are dispatched using [area participation factor control](#) to meet each specific area's determined generation change.

#### Use Area Participation Factors

If the super area control is set to Participation Factor control, you can specify participation factors for each area in the list of areas that indicate how each area should contribute towards changes in generation for the super area. For example, if you have three areas in

the super area and you set the participation factors to 1, 1 and 2, the three areas will contribute to the total change in generation in the super area by ratios of 25%, 25%, and 50%, respectively. The contribution of each area is then further divided among the generators within each area by the participation factors of the individual generators.

This option will be applied when calculating [TLR](#), [PTDF](#), [voltage](#), and [ATC](#) sensitivities. When calculating these sensitivities, the assumption is that the super area is on participation factor control even if the control method is something other than this. If this option is selected, it will be enforced regardless of the control method when calculating these sensitivities.

#### **New Area Name**

Use the dropdown box to select an area to add to the super area. Click the **Add New Area by Name** to add the selected area to the super area.

#### **New Area #'s**

Enter in a list of area numbers separated with dashes or commas and click **Add New Areas by Number** button to add the areas to the Super Area

### **Summary Information**

#### **ACE Total (MW)**

The total area control error for the super area. It is calculated by summing the ACE for all areas comprising the super area.

#### **ACE Tolerance (MW)**

The Area Control Error tolerance observed for the super area during a load flow solution.

#### **Total Scheduled Transactions**

Lists the total scheduled import or export for the super area. It is computed by summing the scheduled interchange for all areas comprising the super area. Exports are positive.

#### **Lambda**

The marginal cost associated with the super area. Lambda is valid only for super areas that are on economic dispatch control.

#### **Hourly Cost**

Current average hourly cost for the super area in \$/hour.

#### **Load, Generation, Shunts, Losses, Interchange**

The Load and Generation section of the Super Area Information Dialog accounts for the real and reactive power flows into, out of, and within the super area. Each quantity is computed by summing over the areas comprising the super area.

### **OPF**

The OPF Tab contains information regarding the [Optimal Power Flow \(OPF\)](#) solution data for a super area. It houses the following information:

#### **Average LMP for Area**

The computed average locational marginal price of all buses contained in the area.

#### **LMP Standard Deviation**

The standard deviation of the locational marginal price for all buses contained in the area.

#### **Min/Max LMP**

The minimum and maximum locational marginal price of all the buses in the area.

#### **Total Generator Production Cost (Scaled)**

The scaled cost includes the Cost Shift and Cost Multiplier. These two values can be defined for each generator, and allow the user a way to assess changes to the LMP results when a generator's cost or "bid" is modified, without actually changing the original generator cost or bid curve. The scaled cost function for each generator is equal to:

$(\text{original cost function} + \text{cost shift}) * \text{cost multiplier}$

#### **Total Generator Unscaled Production Cost**

The total unscaled generator production cost, based on the original generator cost or bid curves.

#### **Total Generator LMP Profit**

The profit of the generators in the area based on the Locational Marginal Prices (LMPs) determined by the OPF solution. The profit is determined as:

$\text{LMP Price} * \text{MW Output} - \text{Unscaled Cost Function}$

#### **Cost of Energy, Loss, and Congestion Reference**

Specify a reference for determining the cost of energy, loss and congestion. The choices are Existing loss sensitivities directly, Area's Bus' Loads, [Injection Group](#), or a specific bus.

#### **Custom**

This page of the dialog can be used to enter notes about the super area. Any information entered in the memo box will be stored with the case when the case is saved to a PWB file. Custom fields can also be entered on this page in the form of floating point, integer, and string fields. These custom fields are stored with the super area and can be used for various tasks in Simulator.

---

## Super Area Field Information

---

Super area field objects are used to show different values associated with super areas. This dialog is used to view and modify the parameters associated with these fields. Note that the super area name itself cannot be changed on this dialog.

### Super Area Name

Select the name of the super area for which you are inserting or viewing information of a super area field.

### Find...

If you do not know the exact super area name you are looking for, you can click this button to open the [advanced search engine](#).

### Total Digits in Field

Total number of digits to show in the field.

### Digits to Right of Decimal

Number of digits to show to the right of the decimal point.

### Field Value

Shows the current output for the super area field. Whenever you change the **Type of Field** selection, this field is updated.

### Field Prefix

A prefix that can be specified and displayed with the selected value.

### Rotation Angle in Degrees

The angle at which the text will appear on the diagram.

### Anchored

Toggles if the field should be anchored to an associated super area object on a diagram.

### Include Suffix

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

### Type of Field

Used to determine the type of super area field to show. The following choices are available:

**Name** Super area name (eight characters maximum). The super area name field must correspond to a valid super area.

**MW Load, Mvar Load** Total MW or Mvar load for the super area.

**MW Generation, Mvar Generation** Total MW or Mvar generation for the super area.

**MW Losses, Mvar Losses** Total MW or Mvar losses for the super area.

**ACE (MW)** Area Control Error in MW for the super area. The super area name field must correspond to a valid super area.

**MW Exports** Total MW exports for the super area.

**Hourly Cost (\$/hr)** The hourly cost for the super area.

**AGC Status** Displays the AGC status of the super area.

**MW Marginal Cost (\$ / MWhr)** MW marginal cost for the super area.



Select **OK** to save changes and to close the dialog, or click **Cancel** to close the dialog without saving your changes.

---

## Interface Information

---

The Interface Dialog is used to create, modify, or delete interface records in both the Edit and Run Modes. This dialog has the following controls:

### Interface Name

An alphanumeric identifier for the interface of up to 24 characters. Use the dropdown box or the spin button to navigate through the list of existing interface records.

### Interface Number

A numeric identifier for the interface, between 1 and 32,767.

### Find Interface...

If you do not know the exact interface name or number you are looking for, you can click this button to open the [advanced search engine](#).

### Add New Interface

Click the Add New Interface button to define a new interface from the Interface Dialog. When you click this button, the **Interface Name** field and **Interface Elements** table are cleared, requiring you to enter a new name and new elements.

### Delete Interface

Select this button to delete the currently displayed interface. Once the interface record has been deleted, the **Interface Name** field displays the previous interface record, if any. If there are not previously defined interface records, Simulator will close the dialog.

### Interface Limits

Specify the possible limits for net interface flow. As for transmission lines and transformers, up to eight distinct limits can be specified for interfaces. Which limit set is used can be controlled from the *Limits Tab* of the [Simulator Options Dialog](#) or the [Line and Transformer Limit Violations Display](#).

### Monitoring Direction

This area allows the user to change the flow monitoring direction. Clicking on the interface in the **Interface Elements** area will bring up the **Interface Elements Dialog**. This is where the default direction is set. To keep this default direction, choose FROM → TO, to change this direction, choose TO → FROM, and to monitor both directions, choose Both Directions.

### Noncontingent MW Flow Contribution

Indicates the present net flow through the interface elements, not including affects of contingencies, in MW.

### Contingent MW Flow Contribution

The amount of flow that would be added to the interface flow if the contingency defined with the interface were to occur, in MW.

### Total MW Flow

The sum of the non-contingent MW flow and Contingent MW flow, for the total MW flow considered on the interface, in MW.

### PTDF Value (%)

The [Power Transfer Distribution Factor](#) for the interface, if calculated.

## Interface Elements

The Interface Elements Table lists each element comprising the interface. If Simulator is in Run Mode, the table will also show the present flow through each element. To edit or delete an existing element in the table, click on it to bring up the [Interface Element Dialog](#). Use the Interface Element Dialog to modify or delete the element.

### Insert New Element

Click the Insert New Element Dialog to add a new element to the interface using the [Interface Element Dialog](#).

Click **OK** to save any changes you have made and to close the Interface Information Dialog. Click **Save** to save your changes but to leave the dialog open so that you can view and modify other interface records. Click **Cancel** to close the dialog without saving your latest change.

### Element Identifiers

Choose whether to show the interface descriptions using the interface names, the interface numbers, or a combination of both.

## OPF

The OPF page will only be visible if you have the Optimal Power Flow add-on for PowerWorld Simulator. This page contains interface information relating to performing an OPF solution.

### Enforce Interface Flow Limit

If checked, the Interface limit will be checked for enforcement during an OPF solution.

### Treat Limit as Equality Constraint

If checked, the OPF routine will try and maintain the flow on the interface at its limit value. Otherwise the limit will be treated as the maximum for an inequality constraint.

### MW Flow on Interface

The actual MW flow on the interface.

### Present MW Limit

The currently defined limit for the interface.

### Percentage of Limit

The MW flow on the interface as a percentage of the limit.

### Limit Marginal Cost

The incremental cost of maintaining the flow on the Interface at its limit value.

### MW Flow Constraint

The constraint status of the interface in the OPF solution is shown here with the corresponding boxes checked by Simulator. These boxes can not be changed by the user.

---

## Interface Element Information

---

The Interface Element Dialog is used to redefine or to add the individual elements comprising an [interface](#). Individual lines or transformers, inter-area ties, inter-zone ties, line contingencies, DC lines, injection groups, generators, and loads may make up an interface. The Interface Element Dialog allows you to add all three varieties of interface elements to an interface.

The Interface Element Dialog comes in two very similar forms, depending upon how it was invoked. The dialog may be called from the [Interface Information Dialog](#) by clicking on either the Interface Elements Table or the Insert New Element button.

The Interface Element Dialog contains the following controls:

### Element Type

Specifies the type of interface element being investigated or added. Interface elements can be any line or transformer or, more specifically, they can be groups of lines and transformers that tie two areas or two zones together. Interfaces can also contain line contingencies, DC lines, generators, loads, and injection groups.

When the selection for Element Type changes, the available **Element Identifiers** change to allow you to pick the appropriate elements of that type.

### Element Identifiers

Depending upon the **Element Type** selection, different element identifiers are required to designate the element to add to the interface. You can search through the list of identifiers to find the particular elements you wish to include. Note that the flow direction on transmission line, transformer, and DC line elements will be dependent on which end of the line you choose as the near bus. The flow will always be measured in the direction of near bus to far bus. The flow will be positive when flowing from near to far, and negative when flow from far to near. In addition, line elements also have an additional setting labeled **Monitor Flow at To End**. This determines which magnitude of flow should be reported. If checked, the flow at the To end of the line will be reported, otherwise the flow magnitude at the From end is used.

### Insert

If you came to the Interface Element Dialog by pressing the Insert New Element button on the Interface Information Dialog, only the **Insert**, **Cancel**, and **Help** buttons will be available. Click the Insert button to add the element you have just defined to the list of elements comprising the interface. After you click Insert, the dialog will disappear, and the Interface Elements table on the Interface Information Dialog will contain the element you just added.

### Replace, Delete

If you arrived at the Interface Element Dialog by clicking on an element in the Interface Elements Table of the Interface Information Dialog, the **Replace**, **Delete**, **Cancel**, and **Help** buttons will be visible. Click **Replace** to modify the interface element according to your specifications on this dialog. Click **Delete** to remove the element from the interface definition.

### Cancel

Click **Cancel** to close the Interface Elements Dialog without saving your changes.

---

## Interface Field Information Dialog

---

Interface field objects are used to show the different values associated with [interface records](#). This dialog is used to view and modify the parameters associated with these fields. The dialog has the following fields:

**Interface Name**

Case insensitive name of an existing interface (12 characters maximum).

**Find...**

If you do not know the exact interface name you are looking for, you can click this button to open the [advanced search engine](#).

**Total Digits in Field**

Total number of digits to show in the field.

**Digits to Right of Decimal**

Number of digits to show to the right of the decimal point.

**Rotation Angle in Degrees**

The rotation angle at which the text field should be displayed.

**Anchored**

If checked, the interface field is anchored to its associated interface display object.

**Include Suffix**

If the *Include Suffix* checkbox is checked, the corresponding field units will be displayed after the current value. Otherwise, only the value without units will be shown.

**Type of Field**

Type of field to show for the interface record. Interface fields may show the interface's name, its MW Limit, MW Flow, percent loading, or any other interface specific field available by clicking the Select a field option and choosing a field from the drop-down list.

---

## Interface Pie Chart Information Dialog

---

Interface Pie Chart objects are used to graphically show the percentage flow associated with [interface records](#). This dialog is used to view and modify the parameters associated with these fields. The dialog has the following fields:

### Interface Name

Case-insensitive name of an existing interface (12 characters maximum).

### Size

Size of the pie chart. Note that the pie chart's size and color can be set to change automatically when the interface's loading is above a specified limit. Please see [Online Display Options](#) for details.

### Actual Value (read-only)

In Run Mode, this field shows the current MW loading for the interface.

### Percent (read-only)

In Run Mode, this field shows the current percentage loading for the interface; if the MVA rating is zero, the percentage is defined as zero, as well.

### Anchored

If checked, the interface pie chart object is [anchored](#) to its associated interface.

### MW Rating

MW limit for the interface, using the current limit set for the case.

### Style

The style of the pie chart. It can be set individually for the particular pie chart, or it can use the style specified in the [Interfaces Pie Chart Options](#).

---

## Automatically Inserting Interfaces in Case

---

The **Inserting Interfaces Dialog** is used to insert a group of [interfaces](#). Only interfaces between adjacent areas or adjacent zones can be inserted automatically; single-branch interfaces between buses must be inserted with line insertion options. Adjacent areas or zones are those that share at least one tie line. To reach this dialog, first open the Interfaces Display by selecting **Aggregations > Interfaces** from the [Model Explorer](#), and then from the [case information toolbar](#) select **Records > Auto Insert Interfaces**. This dialog is NOT brought up through the menu item **Auto Insert > Interfaces** found on the [Quick Insert](#) ribbon group of the [Draw](#) ribbon tab, which is for inserting [interface objects](#) on an oneline diagram.

The Inserting Interfaces Dialog sports the following fields:

### Type of Interfaces to Insert

Select the type of interfaces to insert. Area-to-area interfaces join adjacent areas, while zone-to-zone interfaces join adjacent zones. The name of the new interface defaults to "Area1- Area2" or "Zone1-Zone2" with an **Optional Prefix**.

### Optional Prefix

This field allows you to specify an optional prefix of up to three characters. Use this prefix to avoid duplicating names, particularly when some of the areas or zones have the same name.

### Delete Existing Interfaces

If this option is checked, then all existing interfaces are deleted before inserting the new interfaces. By default, this option is checked. If this option is not checked, the existing interfaces are not deleted. However, new interfaces will automatically overwrite any existing interfaces having the same name.

### Only Insert Between Areas/Zones with Area/Zone Filters Set

If this option is checked, the set of potential areas or zones for inserting interfaces is limited to those for which the area/zone filter setting is *Yes*.

### Limits

Simulator can either calculate an interface rating based on the ratings of the components included in the interface, or the user can specify a set of ratings to be used for the interface. If neither of these options is used to set an interface limit, then by default, the interface limits are left as 0, indicating no limit has been applied.

### Insert Interfaces

Click this button to insert the interface records into the case.

### Cancel

Closes the dialog without modifying the list of interfaces.

---

## Nomogram Information Dialog

---

The Nomogram Dialog is used to create, modify, or delete nomogram records in both the Edit and Run Modes. Nomograms are used for combining a pair of [interface objects](#) for the purpose of monitoring a combined flow restriction on the two interfaces together.

This dialog has the following controls:

### Nomogram Name

An alphanumeric identifier for the nomogram.

### Interface A

The first interface forming the interface pair. To add the interface from scratch in the nomogram dialog, click on the **Insert New Element** button. In this manner you will be creating the interface from the individual elements, the same as creating an interface on the [interface dialog](#). If you already have an interface defined, and wish to clone the elements of that interface for this nomogram, use the button labeled **Clone Elements From** to find the interface and copy the element definition.

### Interface B

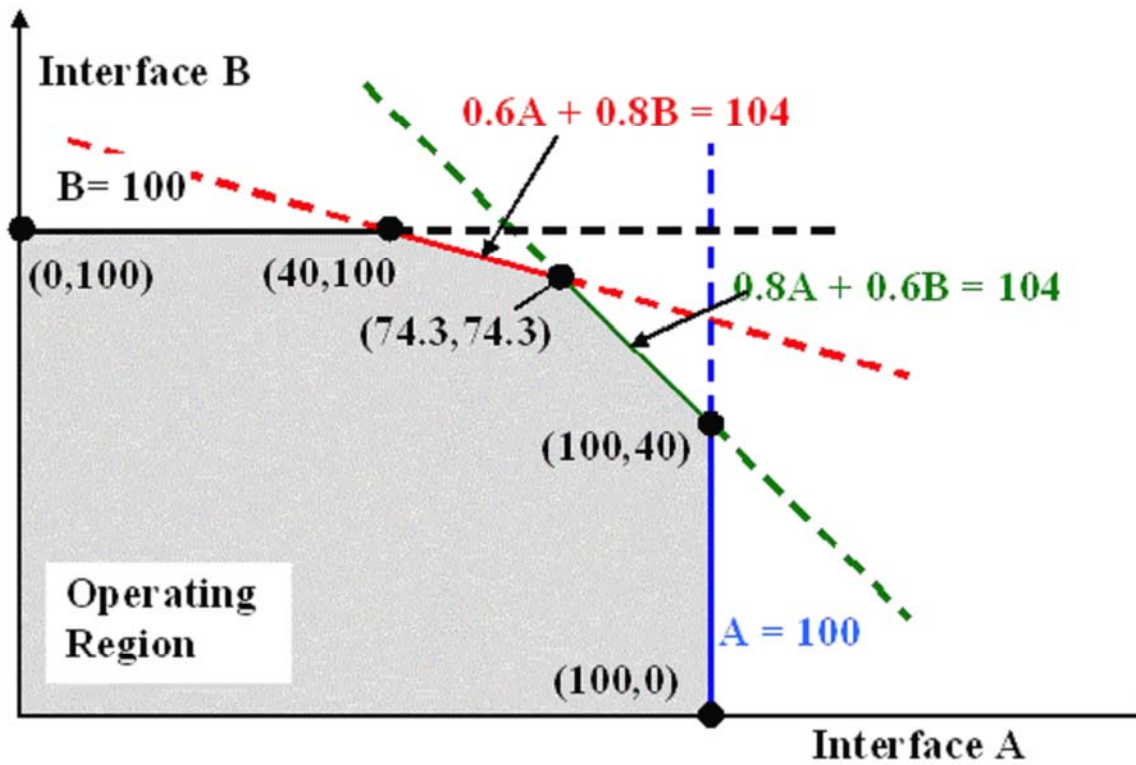
The second interface forming the interface pair. To add the interface from scratch in the nomogram dialog, click on the **Insert New Element** button. In this manner you will be creating the interface from the individual elements, the same as creating an interface on the [interface dialog](#). If you already have an interface defined, and wish to clone the elements of that interface for this nomogram, use the button labeled **Clone Elements From** to find the interface and copy the element definition.

### Nomogram Breakpoints

This section is used for defining the limit boundaries for the interface. The limit boundaries are defined by inserting nomogram breakpoints. These breakpoints correspond to a pair of MW flows on each interface. In other words, you define the amount of flow allowed on Interface B when interface A is at a certain amount. Typically you will have a flow limit on Interface B that is constant for a certain range of flow in interface A. However, at some point as the flow on Interface A increases, the limit of flow on Interface B can start to decrease due to desired flow limit restrictions of the combined interfaces. At some point, the limit of Interface A would reach a maximum amount and remain constant, and the range of flow on interface B would be fairly small due to the heavy loading in Interface A.

To build this Nomogram Limiting Boundary, begin by right-clicking in the Nomogram Breakpoints list and choose Insert Point. Note that the boundary definition must be a convex piecewise linear curve. You would typically begin by defining the flow limit allowed on Interface B when the flow on interface A is small or zero. Then define breakpoints where the limit on B decreases as the flow on A increases. Eventually you will define a point where the flow on A reaches a limit as the flow on B continues to decrease towards zero. In short, the nomogram limiting boundary is actually a combination of boundary limits that are scaled combinations of the individual interface limits. See the image below for an example of a nomogram limiting boundary.





---

## Chapter 8: Text-based Information Displays

---

This chapter describes the case information displays. The case information displays are used to display information about the case, regardless of whether or not it is shown on the onelines.

The following material is included:

- Fundamentals
- Summary Displays
- Area and Zone Displays
- Bus Displays
- Substation Displays
- Generator Displays
- Load Displays
- Line and Transformer Displays
- Switched Shunt Displays
- Interface Displays
- Injection Group Displays
- Island Displays
- Transaction Displays
- Limit Violation Displays
- Ownership Displays
- Load Flow Displays

---

## Model Explorer

---



Most [Case Information Displays](#) are enclosed inside the Model Explorer. To open the Model Explorer, click on the Model Explorer button shown above. The Model Explorer button is available on both of the [Case Information Ribbon Tab](#) and [Tools Ribbon Tab](#) by default. It is also placed on the [Quick Access Toolbar](#) by default. When you open the Model Explorer you will see a case information display. In addition to this the model explorer has 5 additional regions to help you navigate the model as well as modify the model.

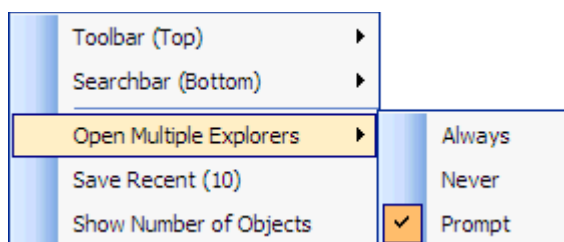
### Explore Pane

The Explore Pane contains a hierarchical list of most of the objects contained in the power system model. Entries in the Explore Pane that are grayed out represent objects that presently are not defined in the model. The Explore Pane is separated into eight primary folders, with only the Network and Aggregations folder expanded by default. The either primary folders are as follows

- **Recent** - contains a list of case information displays that have been recently opened. This same list of case information displays are depicted on the Recent Tabs across the top of the Model Explorer.
- **Network** - contains a list of the network model objects. Generally these represent actual physical devices such as a generator or transmission line.
- **Aggregations** - contains a list of model objects that represent an aggregation of other model objects. An example is an interface which represents such as interfaces or injection groups
- **Solution Details** - provides access to case information displays that provide details regarding the power flow solution. It includes displays such as the Y-Bus, Jacobian, or summaries of remoted regulated buses
- **Case Information and Auxiliary** - contains generic objects that are operate on the other objects in the model. This includes filtering, expression, Model Criteria and such.
- **Contingency Analysis** - contains a convenient place to access case information displays showing options related to contingency analysis.
- **Optimal Power Flow** - contains a convenient place to access case information displays showing options related to the OPF or SCOPF
- **Transient Stability** - contains a convenient place to access case information displays showing options related to Transient Stability

### Explore Pane Options

There are a few options regarding the Explore Pane which can be accessed by right-clicking on the Explore Pane, Search Bar, or Case Information Toolbar, then you will get a local menu that allows you to specify these options



**Open Multiple Explorers** : Choose what to do when you choose to open another Model Explorer if one is already open. You may choose to always open new Model Explorers, Never, or have a prompt window appear each time you request a new Model Explorer

**Save Recent (Number)** : Choose this option to change the number of recently viewed case information that are kept and

shown as tabs in the Recent Tabs

**Show Number of Objects** : Check this option to show a number next to each entry in the Explore Pane

### Options Pane

The Options Pane contains access to the some commonly used case information display actions. All of these options can also be accessed from the [case information local menu](#) or the [case information toolbar](#) as well.

### Recent Tabs

The Recent Tabs represent a list of the most recently viewed case information displays on the Model Explorer. You can click on the X to the left of each tab to clear that from the recent list.

### Case Information Toolbar

The Case Information Toolbar contain a graphical representation of all the options and actions which can be applied to the case information display. The toolbar is described in detail in the [case information toolbar](#) topic. Also note that all the options displayed on the toolbar are also available by right-clicking on the toolbar and showing the [case information local menu](#).

### Search Toolbar

The Search Toolbar contains a toolbar that allows you to search the active case information display for particular text. This toolbar behaves in the same manner as the [Search For Text Dialog](#).

A picture of the Model Explorer as it appears when showing the generator records is shown below. Besides showing case information display,

**Explore Pane**

**Recent Tabs**

**Case Information Toolbar**

**Search Toolbar**

Number of Bus	Name of Bus	ID	Status	Gen MW	Gen Mvar	Set Volt	AGC	AVR	Min MW	Max MW	Min Mvar	Max Mvar	Cost Model	Part. Factor
1	231 BSQSUTH2	1	Open	0.00	0.00	0.99000	YES	YES	0.00	156.00	-112.00	54.00	Cubic	1.99
2	244 WHITNEY1	1	Closed	15.00	3.94	1.02000	YES	YES	0.00	15.00	-5.00	7.00	Cubic	0.17
3	245 WHITNEY2	2	Closed	15.00	3.94	1.02000	YES	YES	0.00	15.00	-5.00	7.00	Cubic	0.17
4	247 BSQSUTH1	2	Open	0.00	0.00	0.99000	YES	YES	0.00	156.00	-112.00	54.00	Cubic	1.99
5	251 BSQSUTH3	2	Closed	155.00	9.80	1.01000	YES	YES	0.00	155.00	-112.00	54.00	Cubic	1.99
6	253 BSQSUTH4	2	Closed	85.00	7.26	1.01000	YES	YES	0.00	85.00	-90.00	40.00	Cubic	1.40
7	332 MILLER1	1	Open	0.00	0.00	1.02000	YES	YES	18.50	75.00	-36.00	48.00	Cubic	0.88
8	333 MILLER2	2	Open	0.00	0.00	1.02000	YES	YES	37.00	120.00	-40.00	73.00	Cubic	1.34
9	334 MILLER3	3	Open	0.00	0.00	1.02000	YES	YES	65.00	208.00	-96.00	115.00	Cubic	2.40
10	335 MILLER4	4	Open	0.00	0.00	1.02000	YES	YES	0.00	104.00	-112.00	54.00	Cubic	1.99
11	336 MILLER5	5	Open	0.00	0.00	1.02000	YES	YES	0.00	110.00	-112.00	54.00	Cubic	1.99
12	344 MORSEHP	1	Closed	12.00	-4.29	1.00200	YES	YES	0.00	12.00	-5.00	6.00	Cubic	0.13
13	345 MORSEHP2	2	Closed	12.00	-4.29	1.00200	YES	YES	0.00	12.00	-5.00	6.00	Cubic	0.13
14	366 TENASKA	1	Closed	168.00	11.64	1.02000	YES	YES	20.00	168.00	-84.00	100.00	Cubic	2.00
15	367 TENASKA	2	Closed	90.00	7.91	1.02000	YES	YES	20.00	90.00	-70.00	68.00	Cubic	1.16
16	503 NTEXAS1	1	Open	0.00	0.00	1.01500	YES	YES	5.00	17.00	-2.90	5.00	Cubic	0.22
17	504 NTEXAS2	2	Open	0.00	0.00	1.01500	YES	YES	5.00	17.00	-2.90	5.00	Cubic	0.22
18	505 NTEXAS3	3	Open	0.00	0.00	1.01500	YES	YES	15.00	39.00	-16.00	15.00	Cubic	0.44
19	580 JKONTY01	1	Closed	300.00	59.94	1.02000	YES	YES	0.00	300.00	-80.00	98.00	Cubic	1.00
20	581 JKONTY02	2	Closed	160.00	23.85	1.02000	YES	YES	0.00	160.00	-40.00	39.00	Cubic	1.00
21	582 JKONTY03	3	Closed	160.00	23.85	1.02000	YES	YES	0.00	160.00	-40.00	39.00	Cubic	1.00
22	719 WFEC 12G	1	Open	0.00	0.00	1.01500	YES	YES	12.00	21.00	-8.00	15.00	Cubic	0.25
23	719 WFEC 12G	2	Open	0.00	0.00	1.01500	YES	YES	12.00	21.00	-8.00	15.00	Cubic	0.25
24	720 WFEC 34G	3	Open	0.00	0.00	1.01500	YES	YES	12.00	20.00	-8.00	15.00	Cubic	0.25
25	720 WFEC 34G	4	Open	0.00	0.00	1.01500	YES	YES	12.00	20.00	-8.00	15.00	Cubic	0.25
26	837 NEWMANS	1	Open	0.00	0.00	1.01000	YES	YES	15.00	40.00	0.00	17.00	Cubic	0.44
27	842 OLINGR1	1	Closed	46.50	15.94	1.03000	YES	YES	20.00	77.00	0.00	40.00	Cubic	0.88
28	843 OLINGR2	1	Closed	63.90	35.34	1.03000	YES	YES	25.00	112.00	0.00	55.00	Cubic	1.34
29	844 OLINGR3	1	Closed	87.10	53.94	1.03000	YES	YES	45.00	146.00	0.00	75.00	Cubic	1.74
30	845 OLINGR4	1	Open	0.00	0.00	1.03000	YES	YES	0.00	87.00	0.00	55.00	Cubic	1.04
31	920 SPNCER1	1	Open	0.00	0.00	1.03000	YES	YES	4.00	12.00	0.00	6.00	Cubic	0.14

---

## Case Information Displays

---

Simulator offers numerous that provide a convenient, spreadsheet-like view of the power system and its components and are available regardless of whether the case has an associated online diagram or not. Many case information displays are available by using the [Model Explorer](#) button found on the [Case Information](#) ribbon tab, although these spread-sheet like views appear throughout the software as stand-alone displays and embedded inside other dialogs. Case information displays are available for buses, bus mismatches, generators, generator costs, ac lines, transformers, transformer impedance correction tables, dc lines, interfaces, areas, zones, schedules, and all other types of objects. Other screens, such as the [Area/Zone/Owner Filters](#) displays to name a couple, also fall into the category of case information displays. There are many different characteristics associated with the Case Information Displays, as discussed in the following topics:

[Colors and Cell Styles](#)

[Using Cell Handles](#)

[Case Information Toolbar](#)

[Sorting Records](#)

[Configuring the Case Information Displays](#)

[Finding Records](#)

---

## Configuring the Case Information Displays

---

The content and format of all case information displays can be controlled through the Display / Column Options dialog. This dialog can be viewed by clicking **Display/Column Options** from the case information display's [local menu](#).

To reset all case information display properties to their default settings, including column and data field associations, click the **Reset All to Default Values** button.

To save your changes and close the dialog box, click **OK**. To save your changes without closing the dialog box, click **Save**. To undo the changes you have specified and close the dialog box, click **Cancel**. Finally, click **Help** to view the corresponding help screen.

### Column Options

#### Available Fields

This list contains all the fields available for addition to the respective case information display. You may select several entries from this list by holding down the Shift and Control keys. Hold down the control key to select several individual fields. Hold down the Shift key to select all fields between two successive mouse clicks. You may then add multiple fields to the Shown fields (see **Add** below).

#### Show these fields in this order

This list contains the fields which are presently shown on the respective case information display. You may select several entries from this list by holding down the Shift and Control keys. Hold down the control key to select several individual fields. Hold down the Shift key to select all fields between two successive mouse clicks. You may then remove multiple fields (see **Remove** below) or move the order of fields (see **Move** below).

#### Column Width

Enter a new value to adjust the physical width of the column. This will affect all fields selected in the Show these fields list.

#### Total Digits

The number of digits to use when displaying values in the column (including the decimal point.) This will affect all fields selected in the Show these fields list.

#### Decimal Places

Number of total digits to the right of the decimal point. This will affect all fields selected in the Show these fields list.

#### < Remove (Removing Columns)

Click the < **Remove** button to remove the fields that are currently selected in the Show these fields list. You may also click on the selected fields in the Show these fields list and drag and drop the fields on the Available fields.

#### Add > (Inserting Columns)

Click the **Add >** button to add all selected available fields to the end of the Show these fields list. You may also click on the selected Available fields and drag and drop the field on the Show these fields list. When drag/dropping the fields, all selected fields will be inserted just before the field that the mouse is over when you drop the fields. Once you have added fields you may then utilize the

#### Move Up, Move Down (Moving Columns)

If you selected a block of fields in the Show these fields list, you may change the order of the list by clicking on the **Move Up** or **Move Down** button.

### **Highlight Key Fields**

Checking this option will highlight the *key fields* for the type of object displayed in the case information display presently being modified. The fields highlighted in Yellow in the two lists are either the Primary key fields (numbered) or the Secondary key fields (lettered). Note that some key fields, usually the Circuit ID, are both numbered and lettered, meaning they are used as both Primary or Secondary key fields. Either ALL numbered key fields OR all lettered key fields must be included in the list of shown columns if you intend to export the data to either Excel or a text file, modify it, and paste it back into Simulator.

The fields highlighted in Green are the fields that would be necessary in Excel or an auxiliary file in order to create NEW objects of the type displayed when pasting from Excel or reading in an auxiliary file. If all of the green highlighted fields are not present in the external source, Simulator will not be able to create new objects that may be defined in that source, and will only paste information for objects that already exist in the present case.

### **Frozen Columns**

The number of columns that are fixed and do not scroll when you scroll left or right in the display. Frozen columns will have a background color of gray instead of white. The number of frozen columns is 1 by default.

### **Show Variable Names**

When this box is checked, the variable name corresponding to each column heading will be displayed in the list of available fields alongside the appropriate column heading.

## **Display Options**

### **Use Area / Zone Filters**

Check this box to restrict the case information display's record set to cover only those areas and zones specified by the [area/zone/owner filters](#). If this box is unchecked, all records, regardless of zone or area, will be displayed. (The area/zone filters option obviously does not apply to the Area/Zone Filters information display.)

### **Advanced Filter**

Clicking this button will open the [Advanced Filters Dialog](#), which will allow more detailed filtering on the display.

### **Use Custom Font / Row Height**

If checked, then the Row Height and Custom Font specifications will be used. Changing either the Row Height or Custom Font properties will cause this box to become checked automatically. If you wish not to apply your custom specifications to the active case information display, uncheck this box; the default font and row height settings, as defined under the *Case Information Displays* tab of the [Simulator Options](#) dialog box, will be reapplied.

### **Row Height**

Defines the height of the rows of the case information display.

### **Change Custom Font**

Defines the font with which to display the records.

### **Set as Default**



Sets the current Font and Row Height as the defaults.

**Auto Size all Column Widths**

Constrains all field widths to contain the widest data elements in each column.

---

## Custom Field Descriptions

---

Custom Field Descriptions allow specification of custom floating point, integer, and string fields with user-defined column headers and field names. These fields can be added to [case information displays](#) for all power system case data and many supporting data types. Custom fields are currently not available for use with case information displays listing display objects. By default, there are two custom fields of each type available, but any number of each type of field can be defined for each type of data.

To view the Custom Field Descriptions, open **Custom Field Descriptions** found under the **Case Information and Auxiliary** folder on the [Model Explorer](#). The table lists the object types for which custom fields have been defined, how many of each type of field has been defined, and any custom field captions and headers for the fields. By default, there will be three entries; there are two fields each for floating point, integer, and string fields for all data types. These defaults will be used unless specific custom fields are defined for a particular data type.

To insert a new description, select **Records > Insert** from the case information toolbar. To modify an existing description, choose **Show Dialog** from the case information toolbar. Performing either of these actions will open the [Custom Field Description dialog](#). To remove an existing description, choose **Records > Delete**. *Default* descriptions cannot be removed.

The following fields are used for defining the custom field descriptions:

### Object Type

Type of object for which the field is available. The *Default* type will be used for any data type that does not have its own definition. This field can only be set using the [Custom Field Description dialog](#) and cannot be changed once a description has been created.

### Field Type

The three options for this field are Floating Point, Integer, and String. This is the type of value that will be used in the custom field. This field can only be set on the [Custom Field Description dialog](#) when inserting a new custom field description.

### Number of Type

This value determines how many fields of the particular Field Type will be available.

### Captions for Field (comma-separated)

Specify the caption to use for each field. These captions will appear in lists of fields for objects such as the list of available fields used with [Display/Column options](#). Enter the captions as a comma-separated list. Leave an empty entry to not assign a particular caption. If a caption is not defined for a particular field, the default will be used. For example, if the Field Type is Integer and there are two fields defined, the Field Captions will be *Integer 1* and *Integer 2*.

If captions are not defined for the fields and Captions for Headers are defined, the header captions will be used instead of the defaults.

### Captions for Header (comma-separated)

Specify the caption to use for each field column header. These headers appear in the case information displays for the particular object type. Enter the captions as a comma-separated list. Leave an empty entry to not assign a particular caption. If a header caption is not defined for a particular field, the default will be used. For example, if the Field Type is Integer and there are two fields defined, the Header Captions will be *Cust Int 1* and *Cust Int 2*.

If captions are not defined for the headers and Captions for Field are defined, the field captions will be used instead of the defaults.

---

**Custom Field Description Dialog**


---

The Custom Field Description Dialog is used to create and modify custom field descriptions for a selected object type. This dialog can be accessed from the [Custom Field Descriptions display](#) when inserting a new description by selecting **Records > Insert** or modifying an existing description by selecting **Show Dialog**.

**Custom Field Description**

Object Name: Bus Find...

Field Type: Floating Point

Number of Type: 3 Clear Captions

	Field Captions	Header Captions
Floating Point 1	My Caption	My Header
Floating Point 2	Another Caption	Another Header
Floating Point 3	Extra Caption	Extra Header

Note: Blank entries for Field and Header captions mean that the default values will be used.

OK Cancel

### Object Name

Type of object for which the field is available. Select the type from the drop-down list or use the Find button to search through a list of available types. This field can only be set when creating a new description and cannot be changed once a description has been created.

### Field Type

The three options for this field are Floating Point, Integer, and String. This is the type of value that will be used in the custom field. This field can only be set when inserting a new custom field description and cannot be changed once a description has been created.

### Number of Type

This value determines how many fields of the particular Field Type will be available.

### Clear Captions

Click this button to clear out any previously defined captions.

### Field Captions

Specify the caption to use for each field. These captions will appear in lists of fields for objects such as the list of available fields used with [Display/Column options](#). If a caption is not defined for a particular field, the default will be used. For example, if the Field Type is Integer and there are two fields defined, the Field Captions will be *Integer 1* and *Integer 2*.

If captions are not defined for the fields and Header Captions are defined, the header captions will be used instead of the defaults.

**Header Captions**

Specify the caption to use for each field column header. These headers appear in the case information displays for the particular object type. If a header caption is not defined for a particular field, the default will be used. For example, if the Field Type is Integer and there are two fields defined, the Header Captions will be *Cust Int 1* and *Cust Int 2*.

If captions are not defined for the headers and Captions for Field are defined, the field captions will be used instead of the defaults.

**OK, Cancel**

Click OK to accept the changes and close the dialog or Cancel to ignore the changes and close the dialog.

## Calculated Fields

Calculated fields provide a means of performing an arithmetic operation (minimum, maximum, sum, average, median, variance, standard deviation) on a group of objects based on a selected field. Based on the object type of the group, a calculated field can be added to a [case information display](#) for a given object. The calculated field value that is displayed is then based on the calculated field parameters and how the object type in the calculated field relates to the given object.

For example, suppose that you want to find the largest line flow for all lines connected to a bus. A calculated field can be set up using the branch object type with a selected field of MVA (maximum) and the maximum operation can be performed on this field. This calculated field can then be associated with buses by adding this field to the [bus case information display](#). The resulting values for this field will show the maximum MVA flow for all lines connected to a given bus. When determining the value, only the branches that have a terminal bus the same as the given bus will be considered, and the maximum MVA value for all of those branches will be reported.

Optionally, filters can be associated with calculated fields to further specify what objects should be considered in the calculation.

The calculated fields case information display can be accessed from the [Model Explorer](#) under **Case Information and Auxiliary > Calculated Fields**. Existing calculated fields can be modified by selecting **Show Dialog** from the [case information toolbar](#), and new calculated fields can be added by selecting **Records > Insert** from the case information toolbar. Both actions will open the Calculated Field dialog shown below.

### Calculated Field Dialog

### Calculated Field Name

This is the name of the calculated field as it will appear when added to a case information display. Use the **Save**, **Save As**, **Rename**, and **Delete** buttons to create new calculated fields without exiting the dialog. Switch between existing calculated fields by choosing a name from the drop-down box.

### Object Type

Type of object that will be included in the calculation. The type of object determines which objects this field can be used with. This information is displayed to the right of the Object Type option.

### Field

Select the field to use in the calculation by selecting a field name from the drop-down box or by clicking the Find button to use [advanced search](#) options to locate the field. Check the **Use Absolute Value** box to use the absolute value of the field in the operation.

### Operation

Select the type of operation to perform on the object fields.

### Treat Blank Entries

Depending on the field selected, some objects may not have entries. Select to either treat these blank entries **As zeros** or to **Ignore** them in the calculation.

### Objects to be included in the operation

This option determines if **All** objects of the selected type will be included in the calculation or if they will be filtered by **Only objects that meet the filter below**.

If choosing to apply a filter, the filter can be defined by using the same options available when defining an [advanced filter](#). The **Set Filter Same As** button can be used to select an existing [advanced filter](#) whose actions should be applied to this filter.

Keep in mind that objects included in the calculation will first be determined by the object type. If for example branch is selected, then all branches will be considered. Setting a filter will limit the branches to only those branches that meet the filter. If the calculated field is used with buses, then the branches considered for a given bus will be only those branches that have a terminal bus the same as the given bus. If the calculated field is used with generators, then the branches considered for a given generator will be only those branches that have a terminal bus the same as the terminal bus of the given generator.

---

### Case Information Customizations Display

---

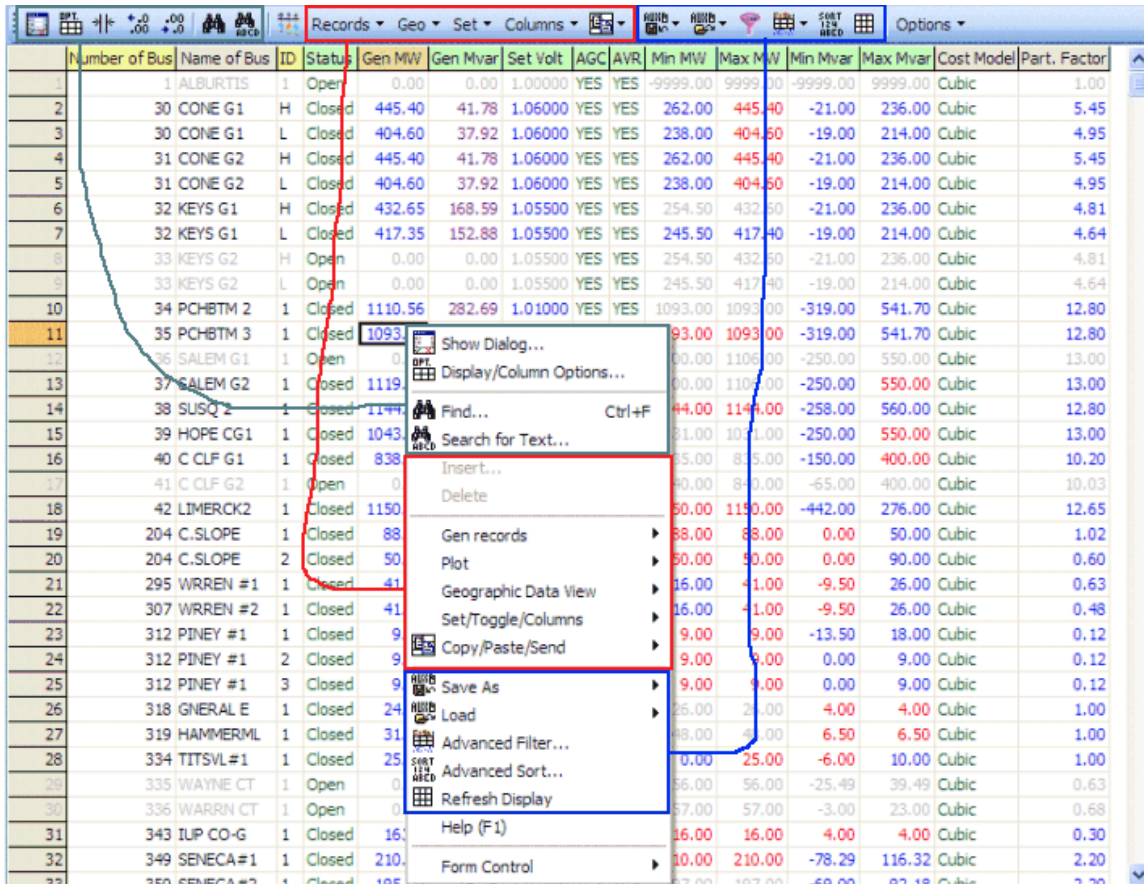
To open the Case Information Customizations display, select **Case Information and Auxiliary > Case Info Customizations** from the [Model Explorer](#). If you have customized any of the case information displays for your case, you can see a list of the customized displays and their settings from this display.

Case Information Customization settings cannot be inserted manually or modified on this display. Those modifications must be done on the actual case information display. The primary use of this list display is to provide you a location to save and load case information customizations to an auxiliary file. Customizations can be loaded from an auxiliary file by right-clicking on the grid and choosing to load data from an auxiliary file. To save the customizations, right-click on this display and select **Save As > Auxiliary Data**.



## Case Information Displays: Local Menu Options

All case information displays have a set of local menu options. The local menu can be brought up at any time by right clicking on case information display. Most of the actions and options available in the local menu are identical to the options available in the [Case Information Toolbar](#). This is depicted in the figure below.



For help on most of the actions and options available in the local menu, please see the [Case Information Toolbar](#) help. The other options that are also sometimes available on the right-click menu however are listed below.

### Why is this field or line disabled?

This option will display a message showing the reason why the field is disabled. This option will be enabled only if the field is actually disabled.

### Help

Display context-sensitive help for the case-information display.

### Form Control

This menu option allows to control the case information display.

### Print

You can print the contents of most of the case information displays by selecting Print from the local menu.

### Close

Closes the case information display.

**Make Top Left of Form Visible**

Makes the top or the left of the case information display visible, if either the top or the left are not visible.

**Shift Form Up**

Moves the case information display up so the bottom of the case information display can be visible.

**Maximize**

Maximizes the size of the case information display.

**Unmaximize (Restore)**

Restores the size of the case information display.

**Minimize**

Minimizes the size of the case information display.

---

## Case Information Displays: Colors and Cell Styles

---

The case information displays distinguish data field types by text color and cell style. Most of the entries on the case information display are colored using the following color convention. These colors can be customized using the Case Information Displays tab on the [Simulation Options Display](#), which can be invoked by selecting **Simulator Options** from the **Case Options** ribbon group on the [Options](#) ribbon tab. **Simulator Options** is also available on the [Tools](#) and [Case Information](#) ribbon tabs.

There are three types of data fields:

**Standard Fields** Fields that cannot be modified directly from the case information display are colored black by default.

**Enterable Fields** Fields that can be modified are colored navy blue by default. When selected, a cell containing an enterable field will display a tiny filled square in its bottom right corner. This box is called a cell.

**Toggleable Fields** Fields whose values can be toggled are colored green by default. The values contained in toggleable fields are modified by left-clicking on them. Like cells in enterable fields, cells in toggleable fields display a cell handle when selected.

Besides indicating field type, color coding is also used to highlight violations of branch flow, generator MW or MVR output, and bus voltage constraints. Fields that are either at a limit or violating a limit are colored red by default. This, too, is configurable from the Case Information Displays tab on [Simulation Options Display](#).

---

### Case Information Displays: Using Cell Handles

---

When selected, cells corresponding to enterable and toggleable fields exhibit a small filled square in their bottom right corner called a cell handle. The cell handle may be used to propagate the value of the selected cell to other cells in the same field.

Suppose we have selected a toggleable or enterable cell and that we wish to copy its value to other records. Call this cell the *source cell* and its value the *source value*. To copy the source value to another record or records, perform the following steps:

- Drag the mouse onto the cell handle until the pointer becomes a crosshair.
- With the mouse pointer showing as a crosshair, click and hold the left mouse button.
- With the left mouse button depressed, drag the mouse up or down from the source cell to select a group of records to which to copy the source value. These destination cells will display a yellow background when selected in this manner.
- When you have finished selecting the destination cells, release the left mouse button.
- A message box will appear asking whether you want to change the values of the destination cells to the source value. Answer Yes to complete the copy.

---

**Case Information Displays: Sorting Records**

---

You can sort the entries on the case information displays by just about any field. To sort the records by a particular column, left-click on the column's heading. Left-click the column's heading again to reverse the sort order.

To sort the records by the absolute value of a field, hold down the shift key as you left-click on the field's heading.

Simulator also has a more advanced sorting tool, which can be accessed by selecting **Advanced Sort** from the [local menu](#). Advanced sort allows you to sort information based on values in more than one column of data. Advanced sort also allows you to sort based on the absolute value of numerical fields and by case sensitivity for string fields.

---

## Case Information Displays: Finding Records

---

The Find Dialog is used to find the device of the specified type on the different Case Information displays. The Find Dialog is available on most case information displays from the local menu, which can be invoked by right-clicking in the grid.

Different Find dialogs exist for Areas, Buses, Interfaces, Lines and Zones. Note that the Bus Dialog is used to locate all bus objects, such as generators, loads, switched shunts and the bus itself. In general, the format of each dialog is similar, allowing you to find the desired object using either its number or name. The only exception is interfaces, for which no name is defined. The basics of the Find dialog are explained in the [Find Dialog Basics](#) help topic.

---

### Saving Case Information Display Contents as HTML Tables

---

Simulator's [Case Information Displays](#) allow you to save their contents as HTML tables for display on the world-wide web. To do this, right-click on any row of the table and select **Save As HTML** from the resulting local menu. This brings up the [Table Format Dialog](#). Set the various table formatting options and click **OK**. Then, select the name of the file to which to save the HTML code. Finally, if a region of the table was selected, you will be asked if you want to save just the selected region as HTML. Indicate **Yes** to convert just the selected portion of the table, or click **No** to write the entire table as HTML.

---

## HTML Table Format Dialog

---

The Table Format Dialog allows you to set various formatting options for HTML tables. This form is invoked when you try to save a [Case Information Displays](#) as HTML.

The Table Format Dialog is divided into two tabs, **Table Properties** and **Table Elements**.

### Table Properties

This tab allows you to specify values for options that govern the appearance of the table as a whole. Here you can specify the following properties:

#### Border Weight

The thickness of the border to draw around each cell. Specify 0 to suppress the drawing of a cell border.

#### Horizontal Cell Spacing

The spacing to employ between cells that neighbor each other horizontally.

#### Vertical Cell Spacing

The spacing to employ between cells that neighbor each other vertically.

#### Table Width

The width of the table. If the Percent checkbox is checked, the width specifies the horizontal dimension of the table relative to the object that contains it on the screen.

#### Caption

The table title that will be printed directly above it on the web page.

### Table Elements

These options control how the data will be centered in each cell. Choices include:

#### Horizontal Alignment

Controls how the text in each cell should be positioned horizontally. The default value is Left, but you may also choose to center or right-justify the data in each cell.

#### Vertical Alignment

Controls how the text in each cell should be positioned vertically. The default value is Middle, but you may also choose to align the text with the top or bottom edges of the cell.

If any of the numeric entries (Border Weight, Horizontal Cell Spacing, Vertical Cell Spacing, Table Width) are left blank or zero, your browser will employ its default settings for these values in rendering the table on the screen.

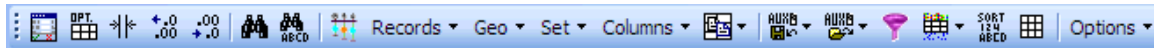


---

## Case Information Toolbar

---

Directly above most case information displays, the Case Information Toolbar provides easy access to many [Case Information Display](#) options for formatting and customizing a list display. Note that all of the options are also available from the [right-click local menu](#) on the case information display as well.



### Show Dialog

Selecting the Show Dialog option will invoke a dialog box containing more detailed information and settings regarding the corresponding system object. For example, clicking Show Dialog while a Bus Case Information Display is the active window opens the [Bus Information Dialog](#).

### Display/Column Options

The contents and format of the information display can be controlled using the Case Information Display Dialog. See [Configuring the Case Information Displays](#) for more details.

### Auto Size all Column Widths

Constrains all field widths to contain the widest data elements in each column.

### Increase/Decrease Decimals

Adjusts the number of displayed decimal places for all cells in the selected column in a case information display.

### Find

Use the Find local menu option to retrieve a record pertaining to a particular element. Choosing Find from toolbar opens the [Find Dialog Box](#), which is used to find records pertaining to an element identified by either number or name.

### Search

Invokes the [Search for Text](#) dialog; allows you to search for specific text in a case information display.

### Bus View Online

On many case information displays click this button to open the [Bus View Online](#) to the bus related to the presently active cell of the case information display.

### Records Menu

The Records Menu contains actions that apply to the specific kinds of records displayed in the active case information display. [Custom Expressions](#), [Bus View Online](#), [Substation View Online](#), [Quick Power Flow List](#) and are always shown in this drop-down. For more details see the [Case Information Toolbar: Records Menu](#) Help.

### Geo Data View Menu

The Geo Menu is only available on case information displays that show records related to the network model. For more details see the [Case Information Toolbar: Geo Data Menu](#) Help.

#### **Set and Columns Menu**

Set contains actions for setting data or toggling data.

Columns contains actions that act on the column such as [Get Column Metrics](#). For more details see the [Case Information Toolbar: Set, Toggle, and Column Menu Help](#).

#### **Copy, Paste and Send Menu**

The Copy, Paste and Send Menu contains actions that copy and paste data through the Windows Clipboard. In addition it contains actions that send data directly to spreadsheet software such as Microsoft Excel or Open Office Calc. For more details see the [Case Information Toolbar: Copy, Paste, and Send Menu Help](#).

#### **Save Auxiliary Menu**

The Save Auxiliary Drop-Down contains actions that save data in the case information display to various formats. The [Auxiliary File Format](#) and Comma-Separate-Variable (CSV) formats are always available, but some records show other more specific formats. For more details see the [Case Information Toolbar Save Auxiliary Files Menu Help](#).

#### **Load Auxiliary Menu**

The Load Auxiliary Drop-Down contains actions that load data records related to the active case information display. The [Auxiliary File Format](#) is always a choice, but some records show other more record-specific formats. For more details see the [Case Information Toolbar: Load Auxiliary Files Menu Help](#).

#### **Area/Zone/Owner Filters**

Opens the [Area/Zone/Owner Filters](#) display which allows filtering of the case information displays by Area, Zone, and Owner.

#### **Filtering Menu**

The Filtering Drop contains actions related to filtering the records shown in the active case information display. For more details see the [Case Information Toolbar: Filtering Menu Help](#).

#### **Advanced Sort**

Allows the user to custom sort the information in the display based on desired criteria. See [Case Information Display: Sorting Records](#) for more information.

#### **Refresh Display**

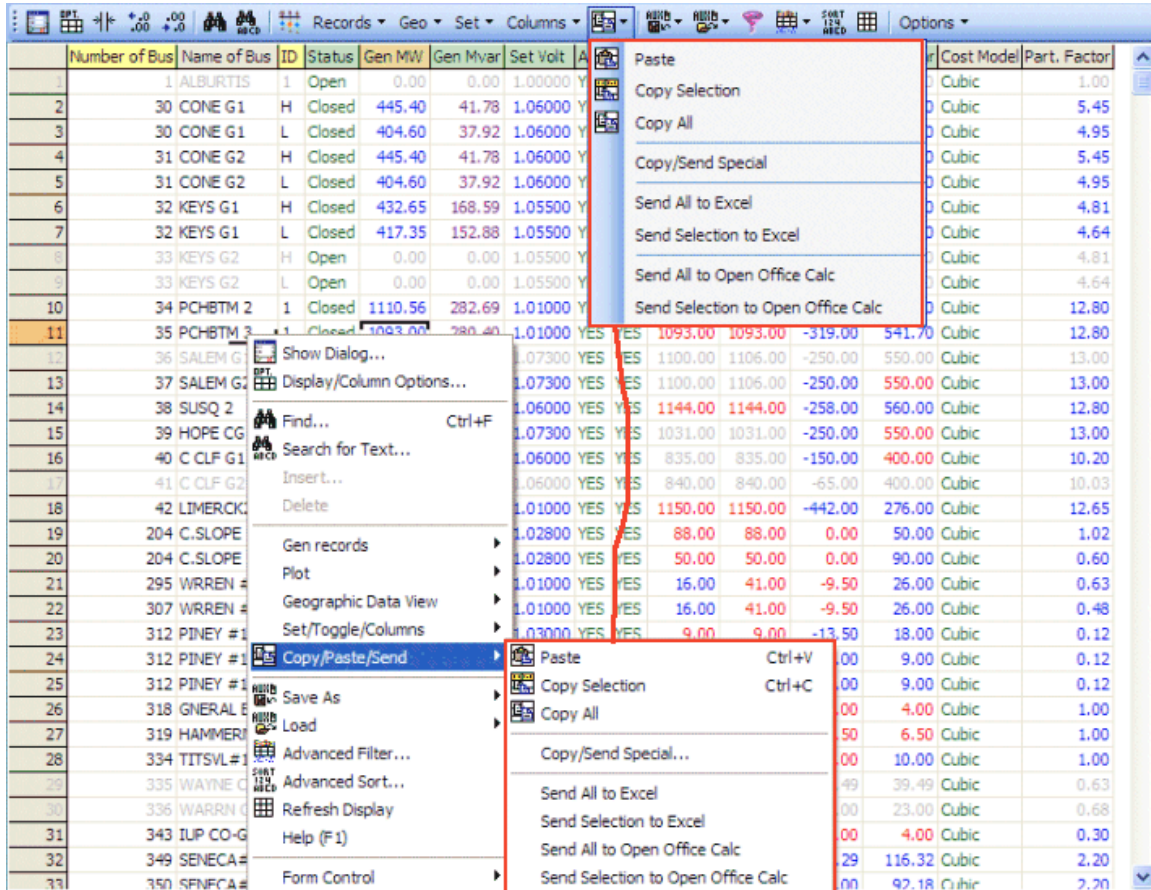
Select this option to update the currently displayed data to match the present state of the system.

#### **Options Menu**

Invokes a drop down menu similar to the Case Information Display options dialog. See [Configuring the Case Information Displays](#) for more details.

## Case Information Toolbar: Copy, Paste and Send Menu

The [Case Information Toolbar](#) Copy, Paste and Send Menu displays the identical options as shown in the Copy/Paste/Send menu of the [case information display local menu](#). This is depicted in the figure below. The options are described below the figure.



### Paste

Select Paste from the local menu to copy a record set from the Windows clipboard into the case information display. See [Copying Simulator Data to and from Other Applications](#) for more details.

### Copy Selection

The Copy Selection menu option copies the records selected in the case information display to the Windows clipboard, from which the selection can be copied into other programs such as Microsoft Excel for further analysis. See [Copying Simulator Data to and from Other Applications](#) for more details.

### Copy All

The Copy All menu option copies the entire record set contained in the case information display to the Windows clipboard, from which it can be copied into other programs such as Excel for further analysis. See [Copying Simulator Data to and from Other Applications](#) for more details.

### Copy / Send Special

The Copy / Send Special menu option will open the a dialog that allows the user to set a few custom options before completing the data copy. The user can choose to copy all or a selection, and to copy the data either to the Windows clipboard or send it directly to Excel. Also, you can choose to copy or send the transpose. In addition, the user can choose whether to use the normal column headings or variable names (see the list of fields in [Data Argument List for Auxiliary Files](#)) as column identifiers. Lastly, the user can specify whether or not to include a row containing the type of object the data represents (Object Name) and a row containing the column headers for each column of data. Note that for pasting the information back into Simulator, the Object Name and Column Heading rows must be contained with the data to be pasted. If you have changed the settings on this dialog and wish to make them the case default settings, click on the Make Default for all Copy Actions button.

#### **Send All to Excel (or to Open Office Calc)**

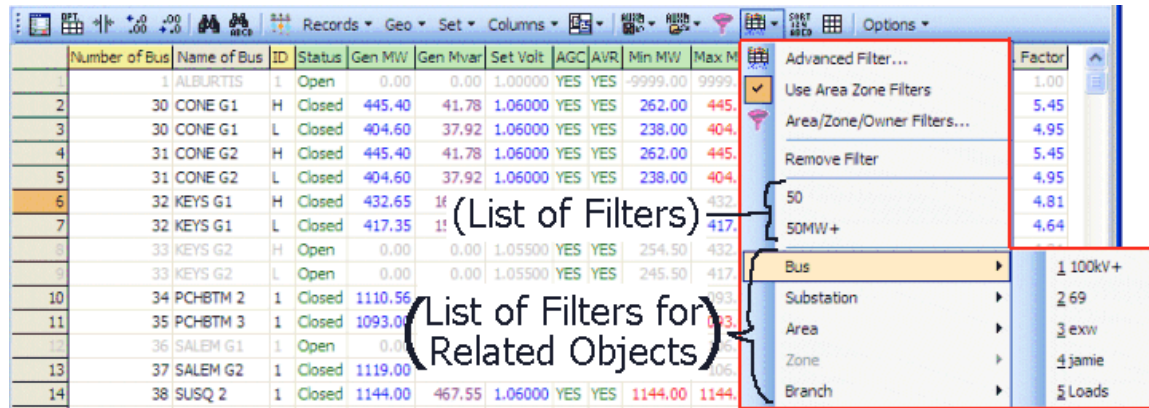
The Send All to Excel menu option copies the entire record set contained in the case information display and automatically sends it to Excel. The first time this option is selected, Simulator will start a new instance of Excel on your machine and paste the data on the first sheet. Subsequent calls to Send to Excel will continue to add sheets and paste data to this instance of Excel, until the Excel instance is closed manually by the user. This functionality works identically for the Open Office Calc spreadsheet as well.

#### **Send Selection to Excel (or to Open Office Calc)**

The Send Selection to Excel menu option copies the selected record set in the case information display and automatically sends it to Excel. The first time this option is selected, Simulator will start a new instance of Excel on your machine and paste the data on the first sheet. Subsequent calls to Send to Excel will continue to add sheets and paste data to this instance of Excel, until the Excel instance is closed manually by the user. This functionality works identically for the Open Office Calc spreadsheet as well.

## Case Information Toolbar: Filtering Menu

The Filtering Menu, contained on the [Case Information Toolbar](#), gives access to several options regarding any [Advanced Filters](#) that can be applied to the active case information display. In addition it provides access to the [Area/Zone/Owner Filters](#). The drop-down menu is shown in the following figure.



The top of the menu has four primary buttons that appear on all case information displays. These four are as follows

### Advanced Filter

Allows the user to custom filter the information in the display based on desired criteria. See [Advanced Filtering](#) for more information.

### Use Area/Zone/Owner Filters

Filters displayed records by selections made on the [Area/Zone/Owner Filters](#) display.

### Area/Zone/Owner Filters

Opens the [Area/Zone/Owner Filters](#) display for setting the areas, zones or owners for which the filter should apply.

### Remove Filter

Click this button to remove the Advanced Filter from the present case information display. This does not delete the Advanced Filter definition, but only removes the reference to it from the active case information display. The filter can easily to reapplied after removing it. If you would like to permanently delete an Advanced Filter see [Advanced Filtering](#).

After the top section of the menu are two sections that may or may not be available for a particular case information display

### (List of Filters)

This contains a list of advanced filters for the type of record displayed in the active case information display. These filters would have already been created using either the [Advanced Filtering](#) dialog or by reading them from an [Auxiliary File](#). In the example image above the "50" and "50MW+" entries represent filters that apply to a generator object.

### (List of Filters for Related Objects)

This contains a list of advanced filters whose object type is *related* to the type of record displayed in the active case information display. For example, the case information display shown in the figure above is showing generator records. Generator records allow you to filter the generator object using a Bus, Substation, Area, Zone, or Branch Filter. In the example above there are 5 filters defined for Bus objects (100kV+, 69, exw, jamie, and Loads) which can be used to filter the generator records. For more information on Related Object Filtering, see the [Advanced Filtering](#) dialog.

---

**Case Information Toolbar: Geo Data View Menu**

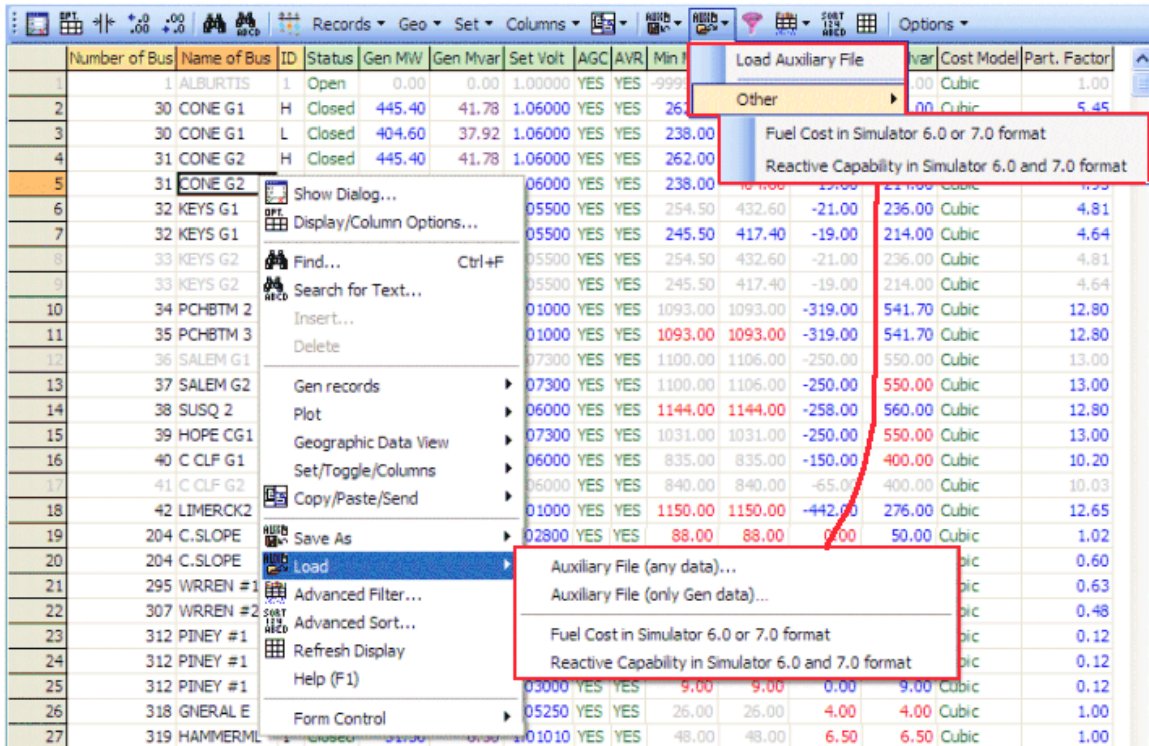
---

The Geo Menu, found on the [Case Information Toolbar](#), contains options related to opening the Geographic Data View Displays based on the objects selected in the case information display. For more information see the help on [Geographic Data Views](#).



## Case Information Toolbar: Load Auxiliary Files Menu

The [Case Information Toolbar](#) Load Auxiliary Menu displays the identical options as shown in the Load menu of the [case information display local menu](#). This is depicted in the figure below. The options are described below the figure.



This option will load the contents of an external file to the case information display.

### Auxiliary File (Any Data)

This option allows loading an auxiliary file containing any data.

### Auxiliary File (Only *Specific* Data)

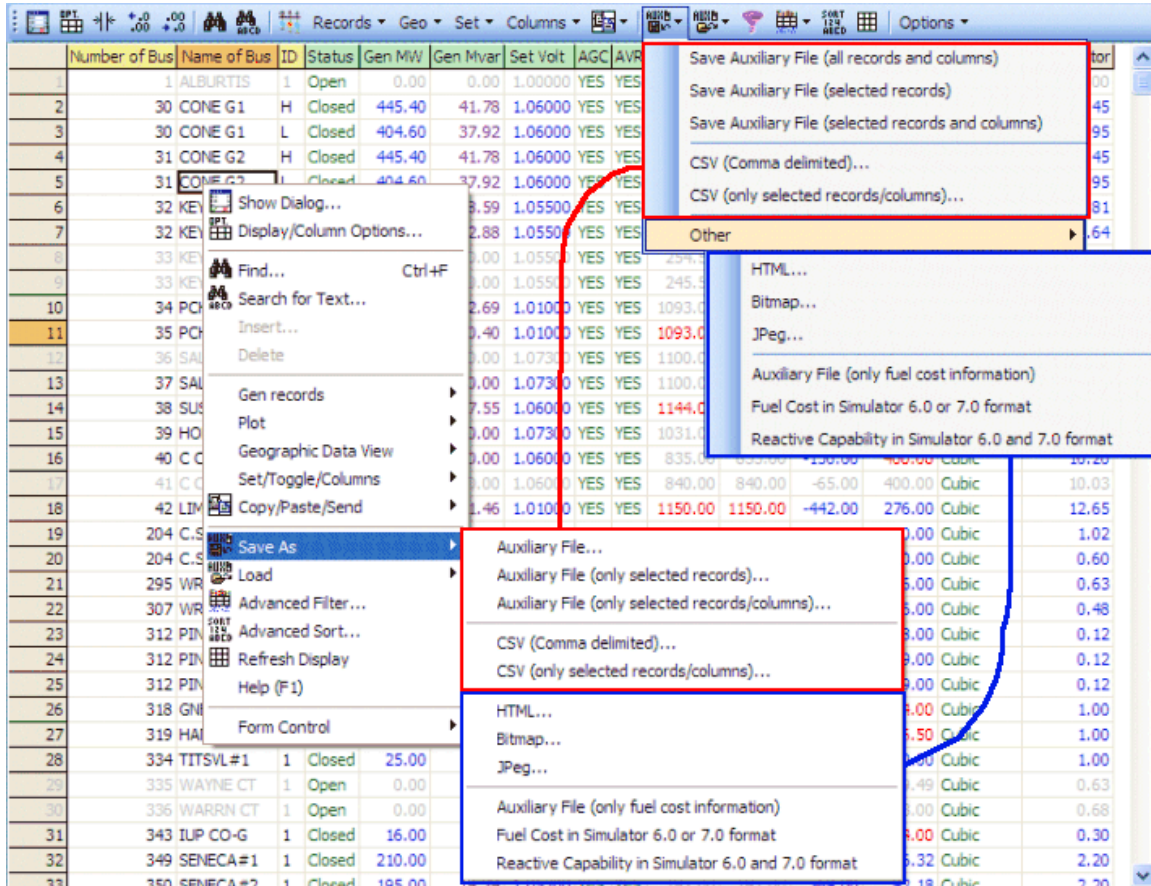
This option allows loading an auxiliary file containing only data related to the current case information display.

At the end of the menu, some case information displays show more specific formats that are relevant only for that type of record. For instance in the figure above you can see support for loading in older file formats previously supported.



## Case Information Toolbar: Save Auxiliary Files Menu

The [Case Information Toolbar](#) Save Auxiliary Menu displays the identical options as shown in the Save As menu of the [case information display local menu](#). This is depicted in the figure below. The options are described below the figure.



This option will save the contents of the case information display to an external file.

### Auxiliary File

This option allows saving all the contents of the case information display to an auxiliary file.

### Auxiliary File (only selected records)

This option allows saving the selected record set in the case information display to an auxiliary file.

### Auxiliary File (only selected records/columns)

This option allows saving the selected record set and the selected columns in the case information display to an auxiliary file.

### CSV (Comma Delimited)

This option allows saving all the contents of the case information display to a comma delimited file.

### CSV (only selected records/columns)

This option allows saving the selected record set and the selected columns in the case information display to a comma delimited file.

**HTML**

You can save the entire table or selected records to an HTML file for viewing from an Internet browser. For more information, see [Saving Case Information Display Contents as HTML Tables](#).

**Bitmap**

This option allows saving the case information display into a Bitmap picture file.

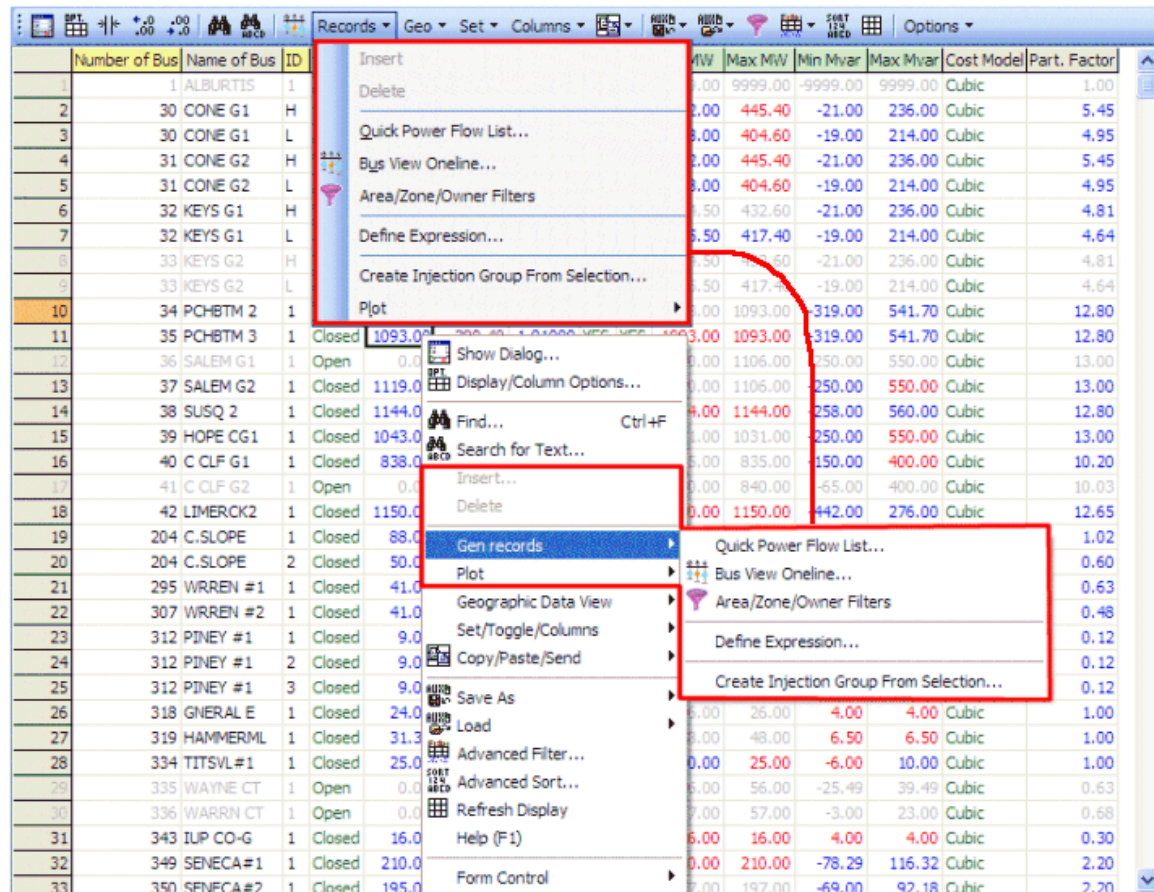
**JPEG**

This option allows saving the case information display into a JPEG picture file.

At the end of the menu, some case information displays show more specific formats that are relevant only for that type of record. For instance in the figure above you can see a specific Auxiliary File format options which automatically saves all the data related to the cost information of the generator.

## Case Information Toolbar: Records Menu

The [Case Information Toolbar](#) Records Menu displays a combination of the Insert/Delete buttons and the Records and Plot menus on the [case information display local menu](#). This is depicted in the figure below. The options are described below the figure.



### Insert and Delete

Many records have an Insert and Delete related buttons. These are used for inserting new records and deleting records. Note that some records can only be inserted or deleted in Edit Mode. In such cases the Insert and Delete button will be grayed out when not in Edit Mode. Use the Bus View Online local menu option

### Records Menu

#### Quick Power Flow List

Use the Quick Power Flow List local menu option to invoke [Simulator's Quick Power Flow List](#) tool.

#### Bus View Online

Use the Bus View Online local menu option to bring up the [Bus View Display](#), which illustrates how the selected bus is connected to the rest of the system.

#### Substation View Online

Use the Bus View Online local menu option to bring up the [Substation View Display](#), which illustrates how the selected substation is connected to the rest of the system.

**Define Expression**

Allows you to define [Custom Expressions](#) that are functions of other fields.

**Pan to Object on Open Onelines**

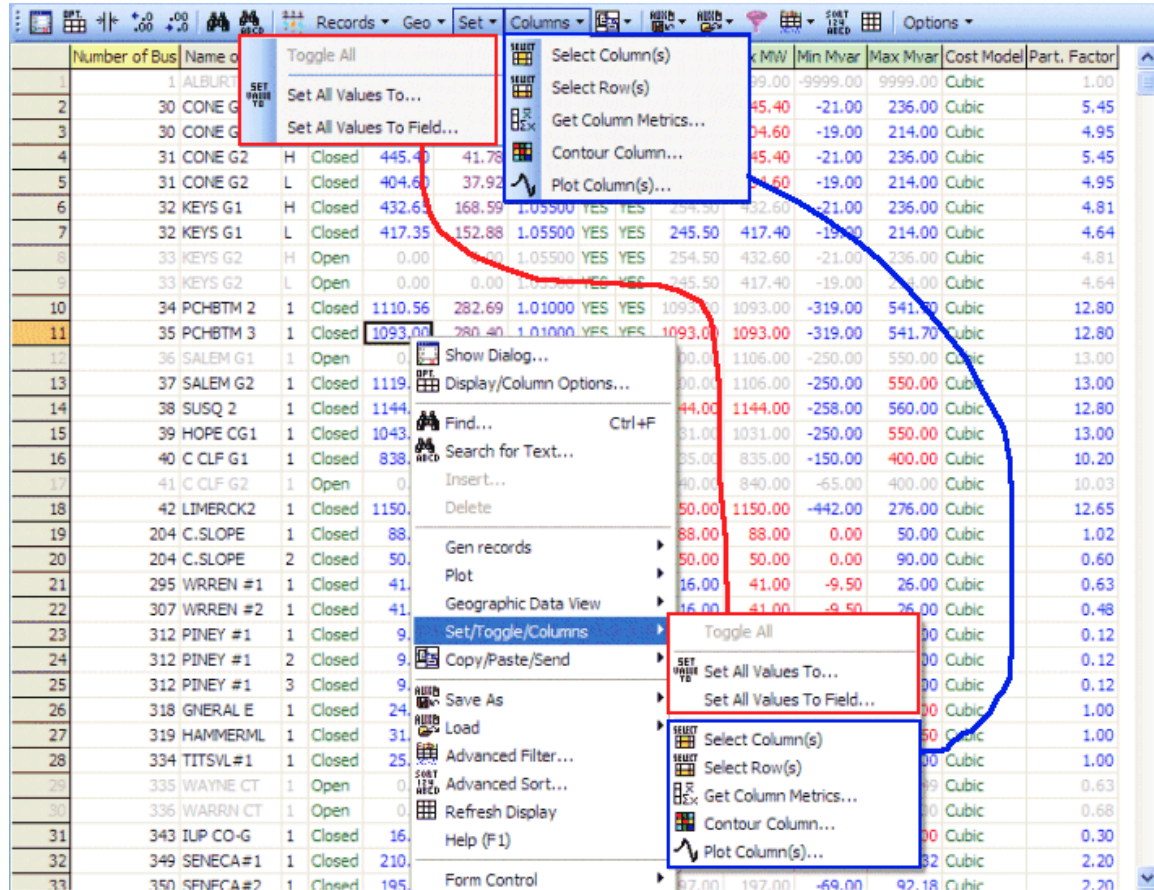
Selecting this option will move the view on any open oneline diagrams to focus on the device clicked in the table, if an object representing the device exists on the diagram.

**Record-Specific Actions**

At the end of the Record Drop-Down will be a list of options that are special for the particular kind of record shown in the active case information display. For example, in the image above for a generator case information display the option "Create Injection Group From Selection..." allows you to create a new injection group containing all of the presently selected generator records.

## Case Information Toolbar: Set, Toggle, and Columns Menus

The [Case Information Toolbar](#) Set and Columns Menu display the identical options as shown in the Set/Toggle/Columns menu of the [case information display local menu](#). This is depicted in the figure below. The options are described below the figure.



### Set All Values To

Click on this menu item to bring up a dialog which allows you to set all the values in a column to the same value. For instance you might change all the buses to have an angle of zero degrees.

A special trick that can be used in conjunction with this option is entering a string of the form "@variablename". When entering data into a field, if the string begins with the symbol @, then Simulator will parse the remaining part of the string to see if it matches one of the variable names associated with that type of object. Variable name strings are what are used in the Auxiliary File Format (see the list of fields in [Data Argument List for Auxiliary Files](#)). Essentially this trick allows you to copy one column to another column. Using this trick, you could change the set-point voltage of all the generators in the case to be equal to the terminal bus voltage of the generator.

### Set All Values To Field

Click on this menu item to bring up a dialog which allows you to select a field of this record type and then set all the values in a column to the this field. For instance you might change all the generator setpoint voltage to be equal to their present regulated bus voltage. This

represents a user-interface method of employing the special trick using the "@variablename" string as discussed in the Set All Value To comments above.

**Select Column(s)**

Use this option to quickly select entire columns in the grid. First highlight the cells of a single record for the columns you wish to highlight, and then choose this option to highlight the entire column.

**Select Row(s)**

Use this option to quickly select entire columns in the grid. First highlight the cells of a single record for the columns you wish to highlight, and then choose this option to highlight the entire column.

**Get Column Metrics**

This option allows you to compute the metrics for the selected column. Choosing Get Column Metrics from the local menu will bring the [Grid Metrics Dialog](#). This option is only available for columns whose content is numeric.

**Contour Column**

Use the Contour Column local menu option to contour a column of the list display. Choosing Contour Column from the local menu will open the [Contour Column Dialog](#).

**Plot Column(s)**

Select this option to plot the values in any selected columns.





---

## Area/Zone/Owner Filters

---

The Area/Zone/Owner Filters Dialog allows you to filter the information shown on the [case information displays](#) and other dialogs by area, zone or owner. There are a several ways to open the Area/Zone/Owner Filters Dialog.

- Go to the [Case Information](#) ribbon tab, and choose  **Area/Zone Filters** from the **Case Information** ribbon group
- Click on the button  on the [Case Information Toolbar](#)
- There are also many buttons on user interface dialogs throughout the program that invoke this dialog as well.

For small cases, you will usually not need to use this filtering capability, but it can be very useful for large cases. The filters display lists each area, zone and owner in the case, the number of buses in each, the range of bus numbers contained in each, and whether or not information about that area, zone or owner should be displayed. In order for a device to be displayed, its area, zone and owner Shown property must be set to yes. For devices which can belong to more than one area, zone, or owner (for instance a transmission line that connects two areas, i.e. a tie line), then the device will be displayed if any end of the device meets the area/zone/owner filter.

You can switch between displaying the filters for the case areas or the zones by clicking on the associated tab.

The area/zone/owners filters list is itself a case information display and therefore shares many of the same [local menu](#) options and characteristics. Using the local menu, you can search for an area, zone or owner by number or by name, copy records to and from other applications, and send the records to a printer. You can inspect an area, zone or owner by selecting *Show Dialog* from the local menu, which invokes the [Area Information](#) , [Zone Information](#) or [Owner Information](#) Dialogs. You can also change the format and content of the filters display by selecting the [Display Column/Options](#) item from the local menu. The records can be sorted by any of its component fields simply by clicking the corresponding column's heading.

The display's only enterable field is the one entitled *Shown*, which may assume only the values Yes and No. For example, if the Area/Zone/Owner Filters setting for an area is *No*, then any case information display configured to enforce area/zone/owner filters will omit the area's elements from the resulting record set. You can specify whether a particular case information display enforces filters using the display's [Display/Column Options Dialog](#).

Double-click on a cell in the *Shown* field to toggle its value. Use the [cell handle](#) to propagate a particular value to multiple areas, zones or owners, or use the *Toggle All Yes* or *Toggle All No* local menu options to set the values of all area, zone or owner records.

---

## Advanced Filtering

---

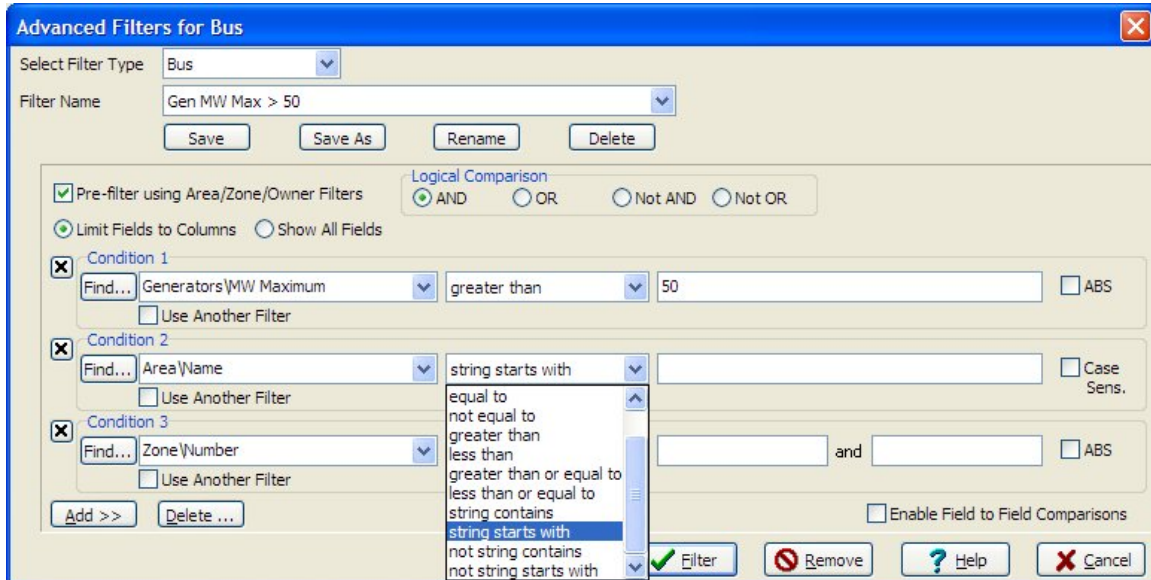
Filtering by areas, zones and owners using [Area/Zone/Owner Filtering](#) is a quick and simple way to filter, however Simulator also contains the ability to perform custom filtering on case information displays as well. Advanced Filtering is accessed by clicking on the [Case Information Toolbar Filtering Menu](#) and choosing Advanced Filter or right-clicking on a [Case Information Display](#) and choosing Advanced Filter from the local menu. This brings up the [Advanced Filters Dialog](#), which allows you to custom filter the information in the display. Advanced Filters are stored with your power system case.

A [Case Information Display](#) will have the phrase "Filter: *Name of Filter*" in its caption if an Advanced Filter has been applied to it. To remove a filter from a [Case Information Display](#), choose Remove Filter from the Case Information Toolbar Filtering Menu or bring up the [Advanced Filters Dialog](#) and click on **Remove**. Note that Remove does NOT delete the filter, but just stops using it to filter the data. You can always remove the filter temporarily and then come back and reapply the filter. If you want to Delete a filter, you must bring it up in the [Advanced Filters Dialog](#) and click on **Delete**.



## Advanced Filters Dialog

The Advanced Filters Dialog is shown in the following figure.



When you open this dialog, you will only see filters that have been defined for the type of object you are trying to filter (e.g. Bus, Generator, Interface, etc...). You can choose a filter from the Filter Name drop-down box showing the list of filters available, or you can create a new filter. To make a new filter, simply click on Save As to save a copy of the present filter under a new name and then specify the properties of the filter as discussed below. When you have specified the filter as you wish, click **Filter**.

Note that advanced filters are stored with the case file when the case is saved. In addition, the filters can be exported to a Simulator [Auxiliary File](#) for storage and import into other cases. A list of all advanced filters defined for a case can be viewed in the [Advanced Filters case information display](#).

To create an Advanced Filter you must specify the following things:

### Select Filter Type

Normally this type is the same as the object type for which you are defining a filter.

However, by changing the drop-down next to Select Filter Type, it is possible to filter based on data for a related object instead of for actual object. For example, an Advanced Filter for a Bus may exist to only show buses only with a Nominal Voltage larger than 138 kV. You may then want to use this filter on a case information display which shows generators. This is possible by changing the Filter Type to be a Bus when showing the Filter Dialog on the Generator case information display. Just click the drop-down to see which related objects' filters can be used for that particular situation.

**Note Regarding Auxiliary Files:** Throughout auxiliary script commands, the ability to specify an advanced filter by name is available. In this context, if the use of a bus filter is desired in a place where a generator filter is needed, then use the string "<BUS> filtername" to represent the filter.

### Filter Name

A string that describes your filter so that you can call it up from other forms in the future.

**Condition 1, Condition2, etc...**

Describes the conditions of your filter. To define a condition

- Specify the field you are filtering. By default, the fields in the drop-down list are limited to the fields presently shown as columns in the display. If you wish to choose from a list of all fields for the object, change the radio button setting to **Show All Fields**. It is very useful to use the **Find...** button to more easily search for the field you are interested in.
- Specify the comparison operation such as "**between**" or "**greater than**".
- Specify the values the field is compared to. Depending on the comparison operation, either one or two values are needed.
- Select **ABS** to use the absolute value of the field value in the comparison operation. For Fields that are strings, select **Case Sens.** to make the comparison case sensitive.

Note: The comparison operation "within integer range list" uses the same format as described in [Entering a Range of Numbers](#).

Note: To add or delete conditions click on the **Add>>** or **Delete...** button. You can also delete an individual condition by clicking on the **X** button next to the condition.

Note: There is no limit to the number of comparisons you use for the Advanced Filter.

**Logical Comparison**

The following describes how the Filter uses the Conditions which are specified

- AND means that all conditions should be true
- Not AND is the opposite of AND (i.e. any one of the conditions can be false)
- OR means that any one of the conditions can be true
- Not OR is the opposite of OR (i.e. all of the conditions must be false)

For advanced logical comparisons within the same filter, refer to the important note below.

**Pre-Filter using Area/Zone Filters**

When this box is checked, data is filtered first by the [Area/Zone/Owner Filters](#) and then by the Advanced Filter, therefore the data must meet both the Area/Zone/Owner Filters and the Advanced Filters in order to be shown. When this box is unchecked, the [Area/Zone/Owner Filters](#) are ignored.

**Enable Field to Field Comparisons**

When this box is checked, it is possible to compare two fields of the device. In order to do so, select **Field** and click on **Find** to select the second field to be compared. The following depicts what a particular condition looks like when the enable Field to Field comparisons checkbox is checked.

The screenshot shows a software interface for defining filter conditions. It features a tab labeled 'Condition 1' with a close button (X). Below the tab, there is a 'Find...' button and a dropdown menu currently showing 'Generators\MW'. To the right of this is a comparison operator dropdown set to 'greater than'. Further right is another dropdown menu showing 'Generators\MW Maximum'. To the far right is an unchecked checkbox labeled 'ABS'. Below the first dropdown is a 'Use Another Filter' checkbox. At the bottom left are 'Add >>' and 'Delete ...' buttons. In the center, there is a 'Field' dropdown menu with 'Constant' and 'Field' options; 'Field' is currently selected. To the right of this dropdown is a 'Find' button. At the bottom right is a checked checkbox labeled 'Enable Field to Field Comparisons'.

**Important Note:** Once you choose a condition comparison type, that type is used for all conditions in the filter. Therefore if you wish to use the AND condition, all conditions you define will be applied using AND. There is a way to combine different conditions within the same filter. This is accomplished by allowing nested filters. In other words, one condition of a filter can be that another filter is met. To refer to one filter from within another, check the box **Use Another Filter**. Then choose whether the condition is to meet the filter or **Not** meet the

filter, then choose the filter (which must have been previously defined) from the drop down list of the third box, or by clicking the Find button. This allows you to define some conditions using one logical comparison in one filter, and then use that filter to combine those conditions with other conditions using a different logical comparison.

For example, consider the logical comparison of **A and (B or C)**. To replicate this, you would define one advanced filter (AF1) that contains the logic **B or C**. Then you can define an advanced filter (AF2) that uses AF1 as **A and (AF1)**.

---

### Advanced Filters Display

---

The Advanced Filters Display is a [Case Information Display](#) available under the folder *Case Information and Auxiliary* on the [Model Explorer](#). The purpose of this display is to list any [advanced filters](#) you have defined for the current case. Each record will list the name of the filter, the type of object the filter is for, filter logic, and whether or not a pre-filter was used. Advanced filters cannot be created or deleted manually from this display, but they can be saved to an auxiliary file or loaded from an auxiliary file by right-clicking on the display and choosing the appropriate Save As or Load option from the popup menu. This feature makes it easy to transfer defined Advanced Filters from one case to another.

---

## Custom Expressions

---

Simulator allows you to define Custom Expressions that are functions of other fields. These Custom Expressions can then be shown as a column in a Case Information Display. To define custom expressions, right-click in a Case Information Display table and choose **Records > Define Expression** from the local menu or choose **Records > Define Expression** from the [case information toolbar](#). This brings up the Define Custom Expressions Dialog. When you bring up the dialog you will only see expressions that have been defined for the type of object shown on the Case Information Display (e.g. Bus, Generator, Interface, etc...).

To define a new custom Expression click **New**.

You can name the expression for easy identification in the list of fields for the object by filling in the **Expression Name** field.

To define the Custom Expression, first specify which fields you would like to use in the expression and assign them to the variables x1, x2, ..., x8. Fields can be chosen that apply to the type of object for which the expression is being defined or a [Model Expression](#) can be chosen. Model Expressions allow using either an expression that applies to model fields or simply a single model field. After fields or Model Expressions have been assigned to variables, type in the expression as a function of the variables x1, x2, ..., x8. For example

$x1 * \sin(x2) + \exp(-x5)$

or

$\tan(x1) + \text{abs}(x6) * 8 - 100$

For a complete list of functions and operators that are available to you, see [Functions and Operators Available](#).

Once the Custom Expression has been defined, you may add the expression to the column of the Case Information Display from which it was called by clicking on **Add Column...** For more information on how to add columns to a display see [Configuring the Case Information Displays](#).

---

## Custom Expressions Display

---

To open the Custom Expressions Display, select **Case Information and Auxiliary > Custom Expressions** in the [Model Explorer](#). If you have defined any [custom expressions](#) for your case, you can see the list of those expressions in this display. The type of object to which the expression applies, the name of the expression, and the custom expression itself are given. The display also shows the values being represented by each of the variables in the custom expression.

Custom expressions cannot be inserted manually in this display, but they can be loaded from an [auxiliary file](#) by right-clicking on the grid and selecting **Load > Auxiliary File (any data)...** from the local menu. Conversely, you can save a list of custom expressions in a case to an auxiliary file by right-clicking on the grid and choosing **Save As > Auxiliary file**.

---

**Functions and Operators Available**


---

Below is a list of functions and operators that are available for use in [Defining Custom Expressions](#):

Symbol	Equivalent	Description	Example
( )		Prioritizes an expression	$5*(1+1) = 10$
!	FACT	Factorial	$5! = 120$ $\text{fact}(5) = 120$
%		Percentage	$35\% = 0.35$
^	**	Raised to the power of	$4^5 = 1024$
*		Multiply by	$3 * 6 = 18$
/		Divide by	$9 / 2 = 4.5$
\	DIV	Integer divide by	$9 \setminus 2 = 4$
MOD		Modulo (remainder)	$7 \bmod 4 = 3$
+		Add	$1 + 1 = 2$
-		Subtract	$9 - 5 = 4$
>		Greater than	$9 > 2 = 1$ * see note
<		Less than	$7 < 4 = 0$
==	=	Equal test	$5 == 4 = 0$
>=	=>	Greater or equal	$3 >= 3 = 1$
<=	=<	Less or equal	$\#h3E \leq 9 = 0$
<>		Not equal	$\#b10101 <> 20 = 1$
NOT		Bitwise NOT	$\text{NOT}(15) = -16$
AND	&	Bitwise AND	$\#b101 \text{ AND } \#h1E = 4$
OR		Bitwise OR	$13 \text{ OR } 6 = 15$
XOR		Bitwise Exclusive OR	$9 \text{ XOR } 3 = 10$
EQV		Bitwise Equivalence	$6 \text{ EQV } 9 = -16$
IMP		Bitwise Implication	$1 \text{ IMP } 5 = -1$
IIF		If condition	$\text{IIF}(1+1=2, 4, 5) = 4$
MIN		Minimum value	$\text{min}(10, 3, 27, 15) = 3$
MAX		Maximum value	$\text{max}(1, 9) = 9$ *see note

SIN		Sine	$\sin(\pi) = 0$ *see note
COS		Cosine	$\cos(\pi) = -1$
TAN		Tangent	$\tan(\pi) = 0$
ASIN		Arc sine	$\text{asin}(1) = 1.570$
ACOS		Arc cosine	$\text{acos}(-1) = 3.141$
ATAN	ATN	Arc tangent	$\text{atan}(0) = 0$
SEC		Secant	$\sec(0) = 1$
CSC		Cosecant	$\csc(1) = 1.18$
COT		Cotangent	$\cot(1) = 0.642$
SINH		Hyperbolic sine	$\sinh(3) = 10.01$
COSH		Hyperbolic cosine	$\cosh(2) = 3.76$
TANH		Hyperbolic tangent	$\tanh(1) = 0.76$
COTH		Hyperbolic cotangent	$\coth(1) = 1.31$
SECH		Hyperbolic secant	$\text{sech}(0) = 1$
CSCH		Hyperbolic cosecant	$\text{csch}(1) = 0.85$
ASINH		Hyperbolic arc sine	$\text{asinh}(2) = 1.44$
ACOSH		Hyperbolic arc cosine	$\text{acosh}(9) = 2.89$
ATANH		Hyperbolic arc tangent	$\text{atanh}(.1) = 0.10$
ACOTH		Hyperbolic arc cotangent	$\text{acoth}(7) = 0.14$
ASECH		Hyperbolic arc secant	$\text{asech}(.3) = 1.87$
ACSCH		Hyperbolic arc cosecant	$\text{acsch}(2) = 0.48$
ABS		Absolute value	$\text{abs}(-8) = 8$
EXP		e to the power of	$\exp(3) = 20.08$
EXP2		2 to the power of	$\exp2(3) = 8$
EXP10		10 to the power of	$\exp10(3) = 1000$
LOG	LN	Natural log	$\log(16) = 2.77$
LOG2		Log base 2	$\log2(8) = 3$
LOG10		Log base 10	$\log10(100) = 2$
CEIL		Round up	$\text{ceil}(6.2) = 7$
RND		Random number	$\text{rnd}(1) = .969$
INT		Truncate to an integer	$\text{int}(6.8) = 6$
SGN	SIGN	Sign of expression (-1, 0, or 1)	$\text{sgn}(-9) = -1$



SQR

SQRT

Square root

$\text{sqr}(64) = 8$

---

## Find Dialog Basics

---

Many times when working with large load flow cases, it can be somewhat difficult to locate devices in the [case information displays](#) regarding a specific device. Simulator has many tools to facilitate filtering data, such as the [Area/Zone/Owner filters](#) and the [Advanced Filtering](#) tool. Even with these helpful tools, finding a device can still be hampered when the bus number or exact spelling of the bus name are not known.

To facilitate locating devices in Simulator, you can use the Find tool to use Simulator's advanced search engine for finding the device(s) you are looking for. The Find tool is available from the popup menus of almost all [case information displays](#), as well as several of the various information dialogs in Simulator. Anywhere you see a button or menu option labeled **Find...** you can open the advanced search tool.

Once the Find dialog has been opened, the dialog will automatically adjust to suit the type of device you are searching for. The caption of the dialog should reflect the type of object the dialog is currently attuned to locate. For most devices, such as buses, generators, loads, etc., there will be one list displayed at the bottom of the dialog containing numbers and names of the type of device you are searching for. In some instances, mostly when searching for branch-type devices, the bottom panel is split with a second list is displayed in the right side panel. This list is used to display the possible connections of the bus selected in the first list. For example, if bus number one is selected and it has connections to bus two and bus three, the second list will display the information for bus two and bus three. Thus you can search for a bus in the first list, then choose from the possible connections in the second list to get a specific branch from the list.

Despite what type of device you are trying to find, the first few options and buttons on the dialog will be the same. **Sort by Name** and **Sort by Number** allow you to choose how you wish to find a device in the list. If you know the bus number you are looking for, choose **Sort by Number**. If you know the name, or at least part of the name, that you are looking for, then choose **Sort by Name**. The list (or first list for branches) will be sorted accordingly.

If you wish to narrow down the list of devices to search through, you have a couple of options for filtering the list before searching through it. First you can make use of the traditional [Area/Zone/Owner filters](#) by clicking in the associated check box. If you need to set more specific conditions for filtering the list, you can instead click the **Define Filter** button to set up an [advanced filter](#). Either one will reduce the number of devices in the list for the search.

Once you have the list set up for your search, you can type in the number or name you wish to find in the text box. If you do not know the exact number or name, you can use wildcards to facilitate the search and find all possible matches for a set of characters or numbers. You can use a question mark (?) to represent a single character wildcard, or an asterisk (\*) to represent a multiple character wildcard.

For example, if you want to find bus number 10005, but all you know is the first four digits are 1000, you can type in \*1000\*, and Simulator will search until it finds the first number that contains those four digits. You can then use the **Search Next** button (note that pressing **Enter** is the same as clicking **Search Next**) to find the next number containing the four numbers, and so on. The same goes for searching by name. If you are looking for bus ACEONE, but all you know is the name contains the string ACE, then you can type in \*ACE\* and then keep pressing **Enter** until you find the bus named ACEONE. By using the beginning and ending \*, we would also find elements such as NEWACE, because the double \* looks for strings that contain ACE anywhere in the string. Note that if you know the first few letters (or numbers), you can narrow down the number of elements found from the search by eliminating the first \* from the search string. For example, to find bus ACEONE, we could have instead used ACE\*, and this would have gone through all matches that started with ACE, ignoring

other elements such as NEWACE. You can also use wildcards in the middle of a string, such as AC\*NE, and Simulator will find any name that starts with AC and ends with NE.

Once you have entered a wildcard search, you can also click the **Search All** button. This will essentially bring up the same Find dialog again, but with the choices narrowed to those that meet your wildcard search.

---

## Search for Text Dialog

---

This dialog allows you to search for specific text in a case information display. Specify the text you want to search in **Search for** edit box. Clicking on **Search Next** will take you to the next field whose content matches with the text specified. The search can be made **By Rows**, in which the text is searched first in all the fields of a record, before searching in the next record. If the search is made **By Columns** then the text is searched first in all the fields of a column, before searching the text in the next column. If the option **Match case** is checked, the search for text will be case-sensitive. The option **Find entire cells only** will take you only to fields whose entire content matches completely with the text you are searching for.

When checking the box **Search Ignore Filters**, Simulator will look through all the records of the type on your present case information display, but will ignore any [Area/Zone/Owner Filtering](#) and any [Advanced Filtering](#) specified with that case information display. If text is found which matches your wildcard search, but is for a record presently filtered out, then you will be prompted to remove the filtering so that the record can be displayed.

When checking the box **Search All Fields**, Simulator will look through every available field which can be displayed for the type of records presently represented on your case information display. If text is found which matches your wildcard search, but is for a field that is not presently shown, then you will be prompted to add the field to the columns that are shown.

---

## Model Conditions Display and Dialog

---

Model Conditions are a type of Model Criteria. Model Criteria represent Boolean expressions regarding the present state of the power system model. They can be used to create a convenient display that shows whether the power system meets a set of criteria. They can also be used in conjunction with the definition of a contingency in the [Contingency Definition Display](#) to create contingency actions that are conditional based on the state of the power system. To see a list of Model Conditions, choose **Case Information and Auxiliary > Model Conditions**.

A model condition contains two parts: a power system element and an [advanced filter](#). The model condition will then return true or false depending on the result of applying the advanced filter to the power system element specified. Model conditions can also be used as part of a [Model Filter](#).

The Model Conditions Display is a type of [Case Information display](#) and has the abilities common to this type of display. To delete a model condition, right-click on the display and choose **Delete**. To insert a new model condition, right-click on the display and choose **Insert**. This brings up the Model Conditions Dialog that can be used to create, delete, and modify Model Conditions.

### Model Conditions Dialog

The Model Conditions Dialog has three sections. The top section provides the ability to **Save**, **Save As**, **Rename** and **Delete** model conditions. To choose a different Model Condition, click on the down arrow next to the Model Condition name.

The middle section provides a location to specify what power system element this Model Condition is related to. On the left is a list of **Element Types** that are available. When clicking on one of these types, the right portion of the dialog will provide you with a list of the elements of this type. This list is a familiar [Find Dialog](#) and provides you the ability to search for the element you are interested in.

The bottom section provides a location to specify an [Advanced Filter](#) which you would like to have applied to the power system element chosen in the middle section. The bottom section of the dialog behaves identically to the [Advanced Filtering Dialog](#). If an Advanced Filter exists for the type of element for which you are creating a Model Condition, you can click the **Set Filter Same As** button to choose this filter. This will then set the parameters of the Model Condition to be the same as the Advanced Filter.

---

## Model Filters Display and Dialog

---

Model Filters are a type of Model Criteria. Model Criteria represent Boolean expressions regarding the present state of the power system model. They can be used to create a convenient display that shows whether the power system meets a set of criteria. They can also be used in conjunction with the definition of a contingency in the [Contingency Definition Display](#) to create contingency actions that are conditional on the state of the power system. To see a list of Model Filters, choose **Case Information and Auxiliary > Model Filters** in the [Model Explorer](#).

A model filter contains a list of [Model Conditions](#) and a Boolean operator to apply to these model conditions. The model filter then returns true or false depending on the resulting of applying the Boolean operator to the Boolean results of the model conditions.

The Model Filters Display is a type of [Case Information display](#) and has the abilities common to this type of display. To delete a model filter, right-click on the display and choose **Delete**. To insert a new model filter, right-click on the display and choose **Insert**. This brings up the Model Filters Dialog that can be used to create, delete, and modify Model Filters.

### Model Filters Dialog

The Model Filters Dialog behaves in a manner very similar to the [Advanced Filtering Dialog](#). To choose model conditions for the model filter, click the down arrows next to the model conditions list. You may also click the **Find...** button to the left of the list. To insert or delete model conditions from the model filter, click the **Add>>** or **Delete** button. When specifying more than one model condition, choose the **Logical Comparison** that will be used to compare the conditions. Finally, the top of the dialog provides the ability to **Save**, **Save As**, **Rename** and **Delete** model filters. To choose a different Model Filter, use click on the down arrow next to the Model Filter name.

To modify the model conditions in the case, click the **Modify Model Conditions Dialog**. This will open the [Model Conditions Dialog](#).

---

## Entering a Range of Numbers

---

On a number of displays it is often convenient to enter a group of numbers, including ranges. Examples include entering buses or areas to scale on the [Scaling Display](#), or buses on the [Quick Power Flow List](#). The format for this field is to enter individual numbers separated by commas, and/or ranges with a dash between the beginning of the range and the end of the range.

For example the entry

1-5,21,23-25

corresponds to numbers 1 through 5, 21 and 23 through 25.

---

## Copying Simulator Data to and from Other Applications

---

You may sometimes find it useful to copy data from Simulator to other applications such as a word processor or spreadsheet program. Alternatively, you may want to copy data from other applications into Simulator. Simulator's [case information displays](#) provide a convenient way to accomplish this. In particular, the **Copy All**, **Copy Selection**, and **Paste** options from the local menu allow Simulator to communicate data with other applications.

To copy a selection of data from a case information display to another application, first select the range of cells to copy from the case information display. Then, right-click on the case information display to display its local menu, and choose **Copy Selection**. To copy the entire content of a case information display to another application, follow the same procedure, except choose **Copy All** from the local menu instead of **Copy Selection**. Switch to the application that will serve as destination for the data and use that application's **Paste** command to finish copying the selected Simulator data to that application. Note that not only is the data copied, but by default so are the data headings. The data headings are very important to maintain if you are planning on copying data from a spreadsheet back into a Simulator case information display. Also note that Simulator copies the text to the windows clipboard with tabs delimiting each cell.

**Note:** Be aware that most programs have limitations on the amount of information you can paste. For example, some spreadsheet programs only allow up to 256 columns and 65,000 rows of information. Some power system information, particularly the Ybus or Jacobian matrices, can easily exceed these limitations.

To paste data from another application into a case-information display, select the data in the other application and use that application's Copy command. In order to paste back into Simulator, you must have the record type and data headings selected with the columns of data. The record type (or object name) must match the case information display you are attempting to paste into, and the data headings (or variable names) tell Simulator which columns of data you are pasting. In order for the data to be pasted to the correct records, you must include the [key field](#) columns in the data to be pasted into Simulator. Also if you are attempting to create new data records by pasting back into Simulator then you must include the [required field](#) columns as well.

It is not necessary to copy and paste all columns or rows of data back into Simulator, as long as the column headings match a valid heading for the case information display. Simulator will skip the data under any column headings that are unrecognized. If you wish to copy data from a spreadsheet into a Simulator display, you must make sure they are ordered in such a way that all the information you wish to paste can be grouped and copied as one block of data from the spreadsheet. Once you have copied all the information from the spreadsheet you wish to transfer, switch to Simulator, open the case information display in which to paste the data, and select **Paste** from its local menu.

**Note:** You can only paste values into the case information displays if the values are enterable on the display (shown blue by default). Also, be careful about pasting redundant data. For example, in the Bus Records display both voltage in *per unit* and voltage in *kV* are enterable, but they specify the same information. Make sure you only copy ONE of these columns into Simulator. Otherwise you may not get what you expect. Simulator will paste the value in twice, and whatever value was pasted 2<sup>nd</sup> will be used.



---

## Save Case Information Data

---

The Save Data Dialog gives you quick access to saving data from certain [Case Information Displays](#) in a PowerWorld Simulator [Auxiliary](#) File.

Specify the name of the auxiliary file in which to save the data records in the text box labeled **Name of File to Save**. Instead of typing the name of the file by hand, you can press the **Browse** button to locate it. Then, indicate whether you want the objects whose data you are writing to the file to be identified by number or by name. Finally, to ensure that only the records currently listed in the [Case Information Display](#) are written to the auxiliary file, check the **Save Only Records Listed in this Display** checkbox. Otherwise, data for all such objects in the entire system will be saved to the file.

When you have finished setting these options, click **OK**. If you changed your mind and do not want to save the data to a file, click **Cancel**.

---

## Key Fields

---

Key fields are necessary fields when attempting to paste data into a [Case Information](#) display from another data application. Other fields are also considered [required fields](#) for the purpose of creating new objects in Simulator by pasting or loading information from another source, such as Excel or a [PowerWorld Auxiliary](#) file. The key fields are easily identified from a few different locations.

### Column Headings

The column headings of case information displays are colored to indicate fields that are [key fields](#). Any field that is a [key fields](#) will be highlighted yellow. Also note that some fields are highlighted green. These fields are [required fields](#) that are necessary when you are attempting to create a new object by pasting or loading the data from another application. If the necessary [key fields](#) (yellow) are not included in the data loaded from another source, the information for that object is ignored. If all of the [required fields](#) (green) are not included in an object that is detected as new, Simulator will skip it. If all the required fields are present, then Simulator can create the new object. Note that for new objects, the key fields are necessary as well.

### Display/Column Options

The Display/Column Options dialog, available by right-clicking on any case information display, also has the capability to highlight both the [key fields](#) and [required fields](#), as described above. The check box labeled "Highlight Key Fields" will enable highlighting of the field names in the two lists on the display.

### Export Object Fields

This option from the Help menu in Simulator allows you to export a list of most fields for each type of object in a case. The list also indicates which fields are [key fields](#) and [required fields](#) for each object. You can output this list of fields as either a text file or into Excel.

The following table lists some of the more commonly accessed key fields needed when attempting to Paste data into existing objects in a Case Information Displays from another data application.

Type of Data	Data Description	Necessary Key Columns for Paste		
Area	Area Records	Area Num		
Bus	Bus Records	Number		
DC Line	DC Line Records	Rectifier Number	Inv Number	Num
Gen	Gen Records	Number	ID	
Interface	Interface Records	Name		
Line/Transformers	Line Records	From Number	To Number	Circuit
Load	Load Records	Number	ID	
Substations	Substation Records	Sub Num		

Switched Shunt	Switched Shunt Records	Number	ID
Zone	Zone Records	Zone Num	

---

## Required Fields

---

Required fields are necessary fields when attempting to create new objects by pasting data into a [Case Information Display](#) or when reading data from an [PowerWorld Auxiliary](#). These fields are in addition to the [key fields](#) which identify the object.

There are two purposes of the required fields. First there are some fields for which an appropriate default value is not apparent thus by requiring these fields it forces the specification of the value. Second, when pasting data into Simulator it is possible that the record will not exist in the case. Simulator will then determine whether to create these records by looking to see if the required fields are specified. By doing this, it prevents the accidental addition of records. For instance you may have an Auxiliary File that sets the Monitor field of all branches in your case to Yes or No. If Simulator runs across a branch between two buses that exist, but whose circuit ID does not exist, then Simulator will just skip this line from the Auxiliary File. It will not be possible to accidentally create a bunch of new transmission lines with default impedance parameters because the R and X impedance parameters are required. The required and key fields are easily identified from a few different locations.

### Column Headings

The column headings of case information displays are colored to indicate fields that are [key fields](#). Any field that is a [key fields](#) will be highlighted yellow. Also note that some fields are highlighted green. These fields are [required fields](#) that are necessary when you are attempting to create a new object by pasting or loading the data from another application. If the necessary [key fields](#) (yellow) are not included in the data loaded from another source, the information for that object is ignored. If all of the [required fields](#) (green) are not included in an object that is detected as new, Simulator will skip it. If all the required fields are present, then Simulator can create the new object. Note that for new objects, the key fields are necessary as well.

### Display/Column Options

The Display/Column Options dialog, available by right-clicking on any case information display, also has the capability to highlight both the [key fields](#) and [required fields](#), as described above. The check box labeled "Highlight Key Fields" will enable highlighting of the field names in the two lists on the display.

### Export Object Fields

This option from the Help menu in Simulator allows you to export a list of most fields for each type of object in a case. The list also indicates which fields are [key fields](#) and [required fields](#) for each object. You can output this list of fields as either a text file or into Excel.

An example of required fields for an object is for a Branch object. For a branch object, the key fields are the from bus number, to bus number and the circuit ID. Required fields are the Impedance Values R, X, and B and the first three ratings A, B, and C. Without specifying values for these fields you can not create a new branch.

---

## Contour Column Dialog

---

The Contour Column Dialog allows you to contour case information display columns.

The Contour Column Dialog has two tabs:

[Contour Type](#)

[Custom Color Map](#)

---

## Contour Column Type

---

### Color Map

Choose from various predefined color maps using the color map combo-box. A color map, along with the values specified, defines how values are mapped to a color on the contour image.

If a color map showing both high and low values is desired (such as for bus voltages), use of "Blue = Low, Red = High" is recommended. If a color map showing only high values is desired (such as for line flows), use of "Weather Radar, Nominal to High" is recommended.

A user may also define additional color maps by going to the [Custom Color Map](#) Tab.

### Reverse Color Map Colors

Check this box to reverse the colors of the selected color map, so the low color becomes the high color, and vice versa.

### Brightness

Modify the brightness track bar to change the brightness of the color map.

### Use cell values directly

Select this option to use the cell values to do the contouring.

### Use the specified field below

Select this option to use a different field value to do the contouring.

#### Value

Select the quantity to use in the contouring from the Value dropdown box or click the **Find Value** button to find the desired field.

### Draw Color Key

Checking this box will cause the contour to draw a color key showing which colors are mapped to which values. You can also give the color key a title, unit label, and specify the number of digits to display in numerical values.

#### Title

Title for the color key.

#### Entry Labels

Units of the contoured value displayed on the color key.

#### Dec. Pts.

Number of decimal places of the contoured value displayed on the color key.

#### Scalar

Multiplication factor that can be applied to the values when drawing the color key.

### Use Equal Spacing For Discrete Maps

This option will draw the color key with equal spacing for all colors in the map, regardless of how close or distant the values the colors represent.

### Use absolute value

Check this check-box to use the absolute values of the quantity selected at the Value dropdown box (above).

### Values

These values along with the color map define how to convert your values into a color for the contour. The values are:

**Maximum** The largest value allowed in the contour. All values above this will be mapped to the highest color. This value corresponds to 100% in the color map.

**Break High** This value is used by some color maps to highlight a lower limit. This value corresponds to 75% in the color map.

**Nominal** This value is the nominal value for the contour. Values around this will be mapped to the middle color. This value corresponds to 50% in the color map.

**Break Low** This value is used by some color maps to highlight a lower limit. This value corresponds to 25% in the color map.

**Minimum** The smallest value allowed in the contour. All values below this will be mapped to the lowest color. This value corresponds to 0% in the color map.

Note: a representation of the color map is shown to the right of the values.

---

### Grid Metrics Dialog

---

The grid metrics dialog allows you to determine the metrics for a set of columns. The metrics determined include: Sum, Average, Variance, Standard Deviation, Maximum, Minimum, Total Items, and Total Non-Zero Items.

By default the dialog is set to determine the metrics for the whole column of the case information display where the option has been selected. However, the selected record set can be modified. The **Start Column** and **End Column** values specify what columns to include in determining the metrics. The **Start Row** and **End Row** values specify what records to include when computing the metrics. If the option **Treat blank cells as zero** is checked, then any blank cells will be considered as zero when computing the metrics. Instead, if the option **Ignore blank cells for calculation** is checked, any blank cell will be taken out of the metrics calculation. Check **Use Absolute Values** to use the absolute values of field values for the metrics computation. Finally, click **Update Metrics** to determine the metrics with the new set of options.



---

## Auxiliary File Export Format Description (For both Display and Power System)

---

### Definition of an Export Format Description

An export format description consists of the following items:

1. A set of object types
2. The fields and subdata to be exported for each object type, including how each field should be formatted when written (e.g., number of digits, number of decimal places)
3. The format (comma-delimited, space-delimited, etc.) that should be used when writing the object data to disk.

For instance, an export format description might have two object types defined—buses and generators—with indications to export the bus numbers for each of these objects and to write this information to a space-delimited .AUX file.

### Types of Export Format Descriptions

Export format descriptions can be defined for both the Display Objects and the Power System Objects,

- Display objects, e.g., background lines, pie charts, displayed fields, etc. Export format descriptions for display objects are defined by selecting Display Objects Export Format Description... on the [Onelines](#) ribbon tab, under the List Display Menu on the **Active** ribbon group.
- Power system objects, e.g., buses, loads, generators, etc. Export format descriptions for power system objects are defined by selecting Power System Objects Export Format Description from the [Case Information](#) ribbon tab on the **Case Data** ribbon group.

The dialogs used to define both types of export format description are very similar; any differences between the two dialogs will be clearly noted below.

### Export Format Description Dialog

Clicking on **Display Objects Export Format Description** or **Power System Objects Export Format Description**, opens the Export Format Description Dialog. There are several buttons at the bottom of the dialog:

- **OK** : save the current settings to the selected export format description, and close the dialog
- **Write Current Format Description to File** : writes data to a file using the current settings. A dialog pops up to select the filename for storing the data. If the file already exists, the data is appended; otherwise, a new file will be created
- **Cancel** : do not save the current settings and close the dialog
- **Help** : open the help associated with the dialog .

The other parts of this dialog box are discussed in detail below

### Format Name Section

The uppermost part of the dialog box is used to manage the set of export descriptions. Once one export description has been saved (by clicking Save or Save As, after defining some object types), a drop-down box will appear which allows selection of an export format description from the set of export format descriptions saved with the case: The buttons underneath the drop-down box have the following functions:

- **New** : clear the rest of the form in order to define a new export format description
- **Save** : save the information in the rest of the dialog box to the export format description currently selected in the drop-down box. If no export format description is selected, this is equivalent to clicking Save As.
- **Save As** : save the information in the rest of the dialog box to a new export format description; a dialog will pop-up asking for the name of the new export format description.

- **Rename** : rename the currently selected export format description
- **Delete** : delete the currently selected export format description

The list of export format descriptions for display objects and power system objects is kept separate, so the drop-down box will only contain export format descriptions defined for the object type selected from the menu (Display Objects Export Format Description Dialog or Power System Objects Export Format Description Dialog).

### File Type and Field Defaults

The next section of the dialog box is used to define the format to save the data, and the default formatting of any numeric fields that are to be saved. Currently, there are 3 options for the file type:

- **AUX (space)** : output the data using a space-delimited .aux file format
- **AUX (,)** : output the data using a comma-delimited .aux file format
- **CSV** : output the data to a comma-separated-values (CSV) file. This type of file can be easily imported into any spreadsheet or database software.

In the **Defaults for Exporting Fields** section, the up/down arrows can be used to change the default number of total digits and digits to the right of the decimal place used in exporting numeric values. These values may be overridden in the Select Fields form for individual fields.

### Object Type and Filter Method Section

The two-column section on the left side of the dialog box is used to define object types and a filter for each object type.

#### Object Type

The first column is used to specify which object types are to be exported. To insert a new object type (corresponding to a new row) into the export format description, there are two options—either click the New button above the **Object Type** column heading, or right click on a row and select **Insert**. Upon doing this, a dialog will pop up to choose an object type:

Type into the text box to search for a particular object type. The list of object types depends on the type of export description being defined. When defining display object export descriptions, only display object types will show up in this list; similarly, when defining power system export descriptions, only power system objects will show up.

Clicking **Choose** will insert the object type into the list of object types to be exported.

Double-clicking an existing object type will pop-up this same dialog box, allowing the object type to be changed. Doing so will empty the exported fields, subdata, and filter method.

#### Filter Method

The second column is used to define an object filter. The default behavior is to export all objects without any filtering, but this can be changed by double-clicking on the filter description. Upon double-clicking on the filter description, a dialog will appear allowing you to choose the filter. The options available for determining which objects are exported are as follows

- **All** : export all objects, disregarding any filters defined in the current case
- **Use Area/Zone/Owner Filter** : only export those objects that satisfy the currently defined area/zone/owner filter. Clicking on **Select Area/Zone** will pop up the dialog box used to define area/zone/owner filters.

- **Use Selected** : only export those objects that have the Selected field set to Yes. Clicking on **Select** pop up the dialog box used to set the Selected flag for each of the objects in the case. Note that this particular option does not exist for display objects, because display objects do not currently support the Selected field.
- **Meets Filter** : only export those objects that meet an advanced filter, which can be either selected from the drop-down box or defined using the **Define Filter** button.

### Fields And Subdata To Be Exported

After specifying an object type (and, optionally, a filter), selecting the row on the left side of the dialog box will allow you to specify the fields and subdata to be exported.

#### Fields To Be Exported

By default, no fields are selected for export. To define fields for export, click on the **Modify** button to the right of the Fields heading the **Select Fields dialog** box will popup. This dialog is used to specify which fields are to be exported, along with any field-specific formatting of numerical values . Generally the dialog behaves identically to the [Configuring Case Information Displays Dialog](#), so for more detailed help see that topic. There are parts of the dialog that behave differently however, and these are described as follows.

- **Total Digits / Dec Places** : these two boxes are used to set the number of total digits and the number of decimal places to use when writing the data for the fields current selected in the Selected Fields section. Leaving these two boxes blank means that the default number of digits and decimal places should be used. Also, if multiple fields are selected that have different settings for the number of digits or number of decimal places, these boxes will be grayed out, but still enterable. Once entering a value into either box, all selected fields will then have the same setting, and the box will no longer be greyed out.

#### Subdata To Be Exported

By default, no subdata is selected to be exported. Also, many objects (e.g., Bus and Area) do not have subdata associated with them, so the **Modify** button will be greyed out. If an object type does have subdata, then the SubData definition section will be grayed out.

To add subdata that should be exported, click the **Modify** button to the right of the SubData heading. This brings up the Select SubData dialog. A description of the options/settings on this dialog follows.

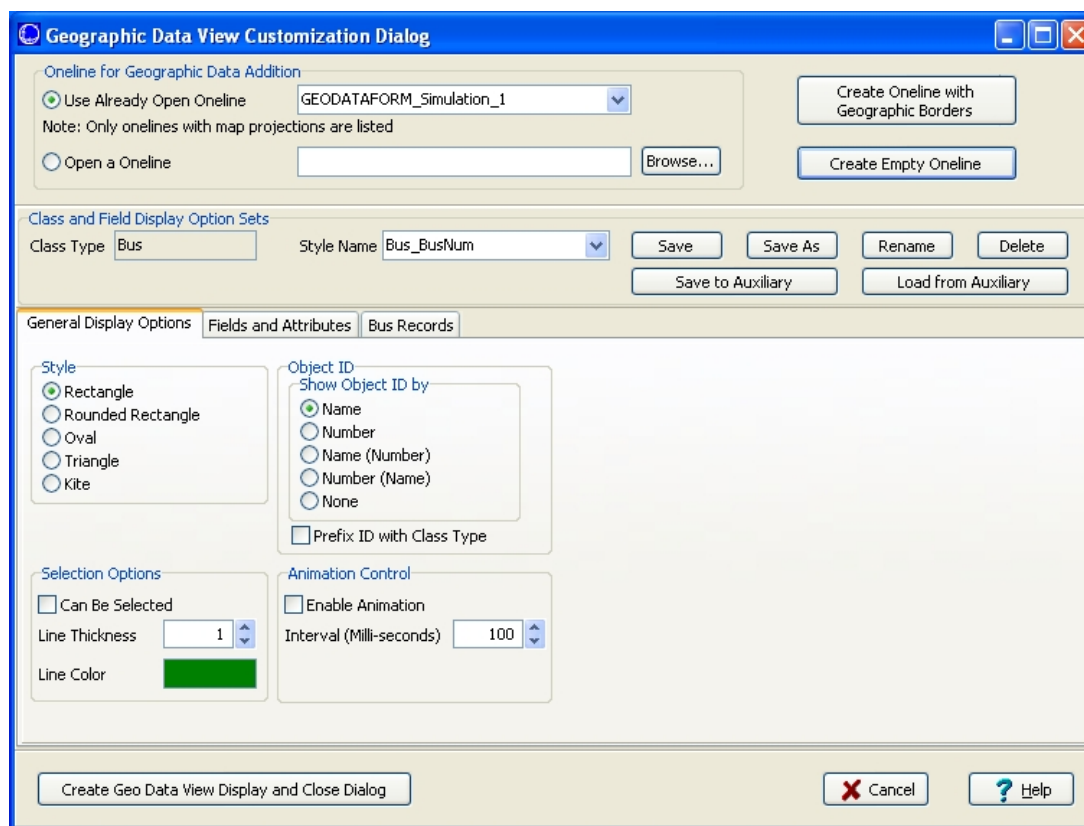
- **Available SubData** : lists the SubData that exist for this object type. If Exclude subdata already selected is checked, then only SubData that is not already in the Selected SubData section will show up in this list.
- **Exclude subdata already selected** : if checked, then the Available SubData section will only contain items that are not already in the Selected SubData section
- **Selected SubData** : lists the SubData to be exported for this object type
- **Add ->** : set the currently selected SubData in the Available SubData section to be exported
- **<- Remove** : set the currently selected SubData in the Selected SubData section to not be exported
- **OK** : save the settings and close the dialog
- **Save** : save the settings, but leave the dialog open
- **Cancel** : do not save the settings and close the dialog
- **Help** : click on this to open the help associated with the Select Fields dialog

## Geographic Data View

Geographic Data Views allow the quick creation and formatting of graphical representations of buses, generators, loads, switched shunts, transmission lines, substations, areas, zones, super areas, and injection groups. Latitude and longitude coordinates specified with buses and substations are used to place objects geographically on a display. The latitude and longitude coordinates must be specified in order to use this feature. Object attributes such as color, size, rotation rate, rotation angle, and visibility can easily be formatted based on the associated data object field values.

To create a Geographic Data View, open a case information display for the desired element type, select the elements for which to add geographic representations, and either right-click on the object grid and choose **Geographic Data View > Geographic Data View** from the local menu or choose **Geo > Geographic Data View** from the [case information toolbar](#). Use the **Select Column, then Geographic Data View** option to select an entire column of elements. The column in which the elements are selected determines the initial selection of the field to use for [attribute settings](#). The Geographic Data View Customization dialog will open.

### Geographic Data View Customization Dialog



### Online for Geographic Data Addition

When creating a new geographic data view, the oneline to which to add the geographic data view objects must be set. Choose either to **Use Already Open Oneline** and then select a oneline from the dropdown list of available onelines or choose to **Open a Oneline** and then either type in the name of a PWD file or click Browse to select a one.

Because geographic data views are dependent on having geographic latitude and longitude information available, the oneline that is selected must have a valid map projection in use. Only those open onelines that currently have a [map projection](#) in use will be listed in the dropdown box and any oneline opened from a file will be checked for a valid map projection. If a valid map projection does not exist, new geographic data view objects will not be added to the oneline.

### **Create Oneline with Geographic Borders**

Click this button to create a new oneline diagram with first being prompted to add geographic borders. A default name will be assigned to the new oneline and that oneline will be added as the selected oneline in the Use Already Open Oneline dropdown.

### **Setting Attributes for Geographic Data View Display Objects**

Before adding new geographic data view objects, the attributes associated with the objects should be set. These attributes are stored in [Geographic Data View Styles](#).

### **Create Empty Oneline**

Click this button to create a new oneline diagram. A default name will be assigned to the new oneline and that oneline will be added as the selected oneline in the Use Already Open Oneline dropdown.

### **Create Geo Data View Display and Close Dialog**

Click this button to accept all of the option settings, add new geographic data view objects to the selected oneline, and then close the dialog.

### **Cancel**

Click this button to close the dialog without adding and new geographic data view objects.

## Geographic Data View Styles: Fields and Attributes

The Field and Attributes options associated with Geographic Data View Styles allow the formatting of the geographic data view display objects depending on the values of fields of their linked data objects.

Attribute	Default for Attribute	Use	Field Lookup
Line Thickness	1	<input type="checkbox"/>	Number
Line Color	Change	<input type="checkbox"/>	Number
Fill Color	<input checked="" type="checkbox"/> Use Fill	<input checked="" type="checkbox"/>	Number
Total Area	<input checked="" type="checkbox"/> Use Auto Width of Objects	<input type="checkbox"/>	Number
Rotation Angle	0.0 (Degrees)	<input type="checkbox"/>	Number
Rotation Rate	0.0 (Hz)	<input type="checkbox"/>	Number
Visibility	<input checked="" type="radio"/> All Visible		
	<input type="radio"/> Only Visible if field is non-zero		
	<input type="radio"/> Only visible if filter is met		
Note: Visibility conditions are not applied in Edit Mode.			
Attributes for All Objects in Style			
Layer	Default Layer	Stack Level	Middle
		Line Dashed	

Fill Color Lookup

Field Set Metrics

Min	1.0000	Median	4.0000
Max	7.0000	Average	4.0000

Color Map

Blue=low, Red=High

☐ Reverse Color Map Colors

☐ Use Discrete Color Map

Dark Normal Lighter

☐ Use absolute value

Maximum	7.00
Break High	5.50
Nominal	4.00
Break Low	2.50
Minimum	1.00

Set Breakpoints from Metrics

### Attribute

Most of the attributes can have a **Default** value or a value that is dependent on the value of a selected field. When the **Use** box is checked for a particular attribute, the field selected in the **Field Lookup** is used to determine the value of the attribute based on either a Color Map or Value Map lookup. Otherwise, the default value will be used. Line Thickness, Total Area, Rotation Angle, and Rotation Rate attributes all use value map lookups. Line Color and Fill Color both use color map lookups. When determining the Field Lookup, either select the field from the drop-down or use the Find button to open the [Find field dialog](#) for easy searching.

When an attribute is selected, either by clicking on one of the options settings for that attribute or just clicking the background box for that attribute, the appropriate color map or value map for that attribute will be displayed in the right-hand panel.

#### Line Thickness

Thickness of the line making up the outside border of the display object.


#### Line Color

Color of the line making up the outside border of the display object.

#### Fill Color

Fill color of the display object. The Fill Color will be used only if the **Use Fill** box is checked.

There are two choices of Field Lookup for this attribute. The top choice is enabled regardless of the choice of object [Style](#). The bottom choice is only enabled when the Kite style is chosen. The Kite style allows two different fields to be displayed in one object. This is useful for displaying fields like Mvar Maximum and Mvar Minimum. The top field indicates what field will be used to format the top half of the Kite, and the bottom field indicates what field will be used to format the bottom half of the Kite.

When using the Kite style, use of the two different Field Lookups can be selected independently and each has a separate color map. When not choosing to use one or both of the Field Lookups, that appropriate half of the Kite shape will use the default Fill Color choice. To switch between color maps for the two Field Lookup choices, either click on the field drop-down for the appropriate half of the Kite or click the  button that will appear next to the field choices when the Kite style is selected.

### Total Area

This determines the size of the display object. Check the option to **Use Auto Width of Objects** for the width of the objects to be determined automatically. If this option is not checked, the **Width** can be set manually. The height or size is always determined as a function of the width and can be set with the **Size to Width Ratio**. When using the Field Lookup table the total area of the object is determined based on the Characteristic Value returned. The dimensions of the object are then determined based on the total area and the Size to Width Ratio.

There are two choices of Field Lookup for this attribute. The top choice is enabled regardless of the choice of object [Style](#). The bottom choice is only enabled when the Kite style is chosen. The Kite style allows two different fields to be displayed in one object. This is useful for displaying fields like Mvar Maximum and Mvar Minimum. The top field indicates what field will be used to format the top half of the Kite, and the bottom field indicates what field will be used to format the bottom half of the Kite. Even though two fields can be chosen when using the Kite style, only one value map is used for the Field Lookup. When using the Kite style and using the Field Lookup the total area of the object is based on the sum of the lookup values returned for both fields.

### Rotation Angle

Static angle of rotation of a display object in degrees.

### Rotation Rate

This value is given in Hz and indicates how many times per second an object should do a full rotation. This is used in conjunction with the [General Option for Animation Control](#). If animation control is not enabled, the rotation rate has no effect on how the objects are displayed.

### Visibility

This attribute can prevent display objects from being shown if certain conditions are not met. Choosing the **All Visible** option will always show all of the display objects associated with this style. If choosing the option **Only visible if field is non-zero**, select a field from the drop-down box. Only those objects whose selected field is non-zero will be visible. If choosing the option **Only visible if filter is met**, select a filter from the drop-down box or click the Find button to search for an existing filter or create a new one. Only those objects that meet the selected filter will be visible. Note that the visibility attribute is only applied while in Run Mode. All objects will be visible in Edit Mode regardless of how this option is set.

### Color Map

Color maps are used with Line Color and Fill Color attributes. The Field Set Metrics section simply gives some statistics on the values of the selected field for the data objects. These metrics are provided to aid in the setting of the breakpoint values for the color map.

### Color Map

Choose from the predefined color maps using the drop-down.

### Reverse Color Map Colors

This checkbox reverses the mapping of the color map, converting the colors corresponding to the high values into the colors for the low values, and vice-versa.

### Use Discrete Color Map

Check this box to make a discrete color map, that is without having smooth transitions between colors.

### Brightness

Modify the brightness track bar to change the brightness of the color map.

### Use Absolute Value

Check this box to use the absolute values of the field values when mapping the field values to colors.

### Breakpoint Values

The breakpoint values determine how the field values will map to colors.

### Set Breakpoints from Metrics

As different fields are chosen for attributes, the Field Set Metrics will be updated to reflect the field values. Click this button to update the breakpoint values based on a new set of Field Set Metrics.

### Value Map

Value maps are used with Line Thickness, Total Area, Rotation Rate, and Rotation Angle attributes. The Field Set Metrics section simply gives some statistics on the values of the selected field for the data objects. These metrics are provided to aid in the setting of the Field Values and corresponding Characteristic Values.

The attribute value for a particular display object is set to the Characteristic Value that corresponds to the field value of the corresponding data object. Characteristic values will be interpolated if the field value does not fall exactly on a value that has been defined in the table. To insert entries into the table, right-click on the table grid and choose Insert from the local menu. Choose Delete from the local menu to remove entries.

Line Thickness Value Lookup

Field Set Metrics

Min	1.0000	Median	4.0000
Max	7.0000	Average	4.0000

Field Value	Characteristic Value
1.000000	1
4.000000	2
7.000000	3



## Attributes for All Objects In Style

These attributes apply to all objects independent of any specific field value.

### Layer

Type in the name of an existing layer to assign the objects to an existing layer, or type in the name of a new layer to have that layer created and add the objects to that layer.

### Stack Level

An object's stack level dictates what objects it will appear above and which objects it will appear below on a oneline display. For example, circuit breaker and pie chart objects have a default stack level of Top. Therefore, anything with a stack level of Middle, Background, or Base will appear underneath pie charts and circuit breakers on the oneline display. Objects that are within the same stack level and are drawn in the same location will result in the last object drawn being the visible object on the display.

### Line Dashed

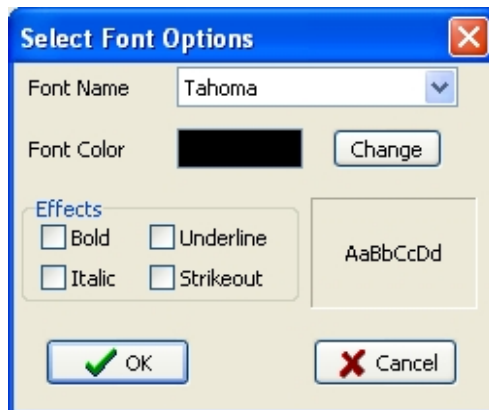
This determines the appearance of the line making up the outside border of the display object. This line can be solid or one of the dashed options.

### Immobile

Check this box to keep the geographic display objects from being moved on the oneline diagram.

### Set Font

Click this button to bring up the Select Font Options dialog. The changes to font are limited to Font Name, Font Color, and special Font Effects of Bold, Underline, Italic, and Strikeout. There is no option for setting the font size because this is automatically determined based on the size of the display object.



---

## Geographic Data View Styles: General Display Options

---

The General Display Options associated with Geographic Data View Styles are applied to all geographic data view objects independent of any field value.

### Style

The style determines the shape of the geographic data view display object used to represent the selected data records.

### Object ID

Each geographic data view display object can be identified by name, number, or a combination of the two. When a choice other than *None* is selected, the appropriate identification for that object will be displayed. When **Prefix ID with Class Type** is selected, the object class type will be prefixed along with the main identifier. This option is useful when trying to distinguish between the different types of objects shown on a given display.

### Selection Options

Geographic data view display objects can be formatted differently when they are selected as opposed to not selected. Check the option **Can Be Selected** to allow objects to be formatted based on the set options when they are selected. **Line Thickness** determines the thickness of the border line around the outside of the display object when it is selected.

**Line Color** determines the color of the border line around the outside of the display object when it is selected.

### Animation Control

Geographic data view display objects are capable of being animated if a [Rotation Rate](#) other than zero is specified with the [attributes](#). Check the **Enable Animation** box to animate the objects using this style. The **Interval** determines how often the display is refreshed during animation.

---

## Geographic Data View Styles

---

Each geographic data view object has an associated Geographic Data View Style. The style determines how the object is going to be displayed and contains options for setting various attributes associated with the object. Each style refers to a particular type of object. Geographic data view objects cannot be individually formatted; they will always obey the formatting of their associated style.

The **Geographic Data View Options** dialog is used to create and modify styles that will determine the appearance of geographic data view objects. This dialog is part of the [Geographic Data View Customization](#) dialog when adding new geographic data view objects to a display, but it becomes a stand-alone dialog when modifying the styles for existing objects. To open as a stand-alone dialog, right-click on an existing geographic data view object and choose **View Geographic Data View Options** from the local menu.

Usually, groups of objects share the same geographic data view style. Keep in mind that changing the options in a style will change all of the objects that use this style.

### Geographic Data View Options Dialog

The Geographic Data View Options dialog is made up of four main sections and tabs:

#### Class and Field Display Option Sets

The screenshot shows a dialog box titled "Class and Field Display Option Sets". It has a "Class Type" dropdown menu with "Bus" selected. To its right is a "Style Name" dropdown menu with "Bus\_BusNum" selected. Below these are two rows of buttons. The first row contains "Save", "Save As", "Rename", and "Delete". The second row contains "Save to Auxiliary" and "Load from Auxiliary".

This section allows the saving, renaming, and deleting of Geographic Data View Styles. Only those styles for a particular **Class Type** will be displayed in the **Style Name** dropdown box. Style names must be unique within a Class Type. Geographic data view styles can be stored in the standard [auxiliary file](#) format. When saving or loading an auxiliary file from this dialog, all geographic data view styles will be stored or loaded regardless of the current class type. The default is to store the information in a data auxiliary file, .AUX, but these objects are also supported in display auxiliary files, .AXD.

#### [General Display Options](#)

#### [Fields and Attributes](#)

#### Object Records

The object records tab lists the data objects that are linked to the geographic data view style, through their associated geographic data view display objects, selected when the dialog is opened. The caption of this tab will be updated to reflect the class type of object that is represented. For example, if viewing bus objects, the caption will be *Bus Records* instead of *Object Records*. These are either the data objects selected before creating a new geographic data view or data objects represented by existing geographic data view objects that are using the selected style. The format of the display objects represented by these data objects will be changed based on any changes made to the options. To assign a group of geographic data view display objects to a new style, simply select a different style than the style in place when opening the dialog and click either the **OK** button on the **Geographic Data View Options** dialog (existing data object was selected) or the **Create Geo Data View Display and Close Dialog** button on the **Geographic Data View Customizations** dialog (new geographic data view objects are being created).

To change the style of a single geographic data view display object and assign it to a new style, use the Display Explorer. Within the Display Explorer, [case information displays](#) for

geographic data view objects are available. The style is given in the field **Geographic Data View Style Name** (variable name GEODATAVIEWSTYLENAME). To change the style, change the name listed in this field to a style that is available for the class of object being changed.

---

## Custom Case Information Display

---

To open the Custom Case Information Displays, go to the [Case Information](#) ribbon tab, and choose **Custom Case Info** from the **Case Data** ribbon group

Custom Case Information Displays can be used to create a display very similar in appearance to a spreadsheet workbook with several worksheets. This display can show any information that can be shown on a [case information display](#), but also allows you to customize the layout of the information in any manner.

There are three buttons which allow you to **Rename** the present sheet, add a **New** sheet, or **Delete** the present sheet.

There are three different kinds of cells allowed on this display

- Blank Cell – contains nothing
- Plain Text Cell – contains a user-entered string with no link to any data in the model.
- Model Field Cell – contains a link to model field similar to inserting a [model field](#) on a oneline diagram

Each cell of the display will behave differently depending on the Custom Case Info Mode. There are four distinct modes that control the user interaction and operation of the custom case information display. These four modes and the effect they have on the display are described in topics

- [Define Fields/Strings](#)
- [Change Field Data](#)
- [Show Fields Primary](#)
- [Show Fields Secondary](#)

## Define Fields/Strings

The "Define Fields/String" mode is the primary mode for use when setting up a new [Custom Case Information Display](#). The three kinds of cells behave as follows in this mode.

### Blank Cell

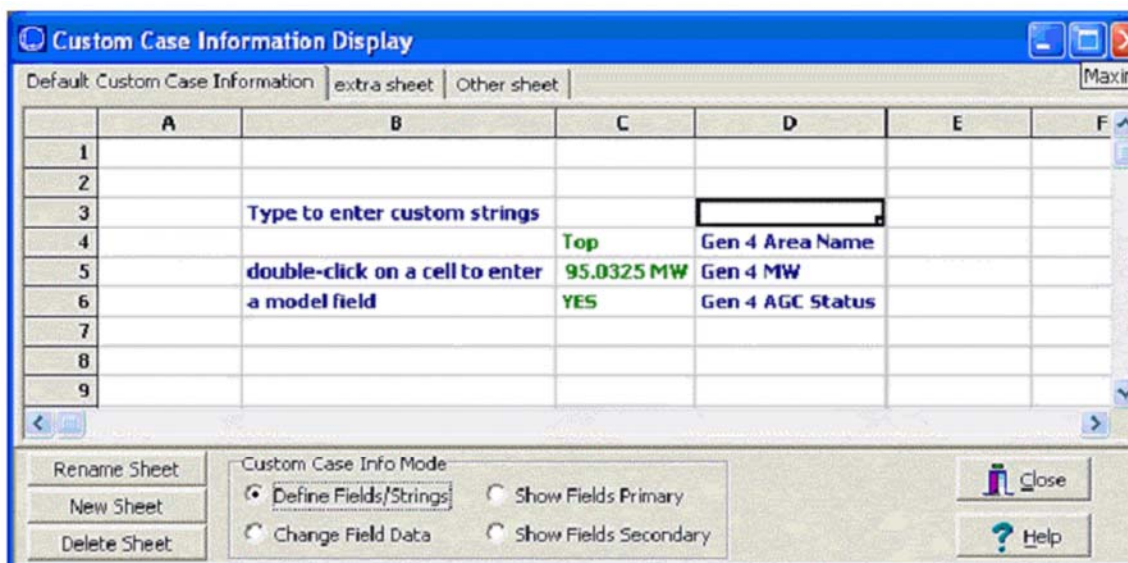
To convert to a Plain Text Cell, just type in the cell. To convert to a Model Field Cell, double-click on the cell to open up a dialog for defining the model field.

### Plain Text Cell

These cells will appear in the Case Info Display Enterable Color (blue by default). To change them just type on the cell. When pasting into such a cell or editing it directly it will parse the string entered trying to create a Model Field as though the string represents the model fields as shown in the Show Fields modes. If the string does not represent such a field, then it will remain a Plain Text Cell

### Model Field Cell

Model Field cells will not be enterable. In order to edit these fields, you must double-click on the cell to open a dialog for defining the model field.



## Change Field Data

The "Change Field Data" mode can be used to edit the data referred to by the model fields on a custom case information display. The three kinds of cells behave as follows in this mode.

### Blank Cell

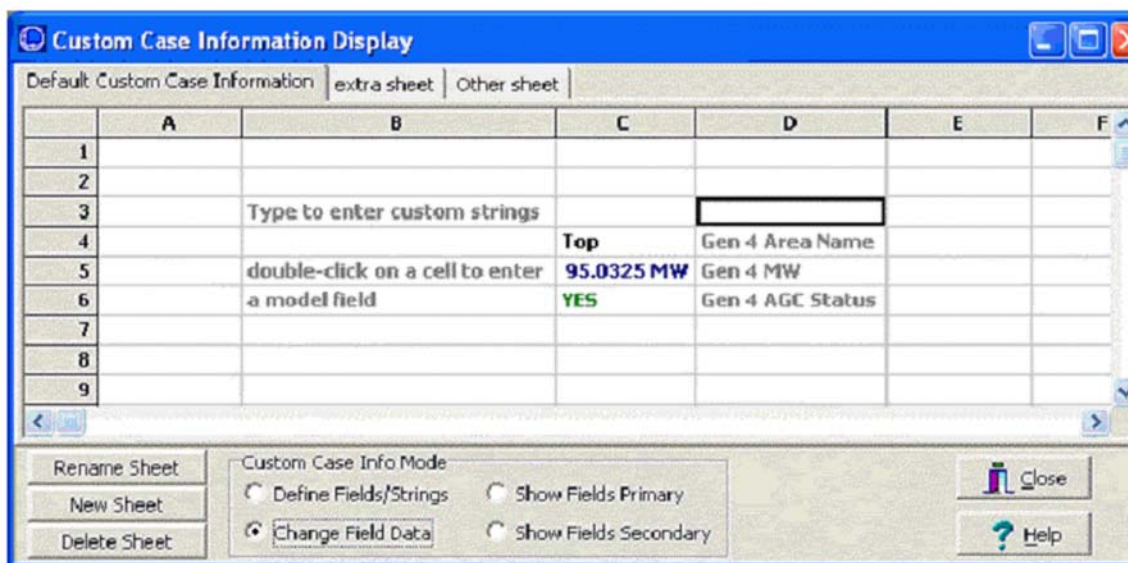
Blank cells may not be edited in any manner.

### Plain Text Cell

Plain Text cells may not be edited in any manner. It will appear in a special color defined for the custom case information display which may be specified from the [local menu](#) of the workbook tabs. By default this color is dark gray.

### Model Field Cell

Model Field cells will behave according to the field to which they refer. Enterable fields will be enterable, toggleable fields will toggleable, etc... When pasting in the sheet in this mode you will be modifying the model data directly. In the following picture there are three fields in cells C4, C5, and C6 that refer to read-only, enterable, and toggleable field respectively. The fields are labeled using the plain text cells in D4, D5, and D6.



---

## Show Fields Primary

---

The "Show Fields Primary" mode is used for interacting with an external spreadsheet to create the model field links in custom case information displays. The three kinds of cells behave as follows in this mode.

### Blank Cell

Blank cells behave the same as for the "[Define Fields/Strings](#) " mode.

### Plain Text Cell

Plain Text cells behave the same as for the "[Define Fields/Strings](#) " mode.

### Model Field Cell

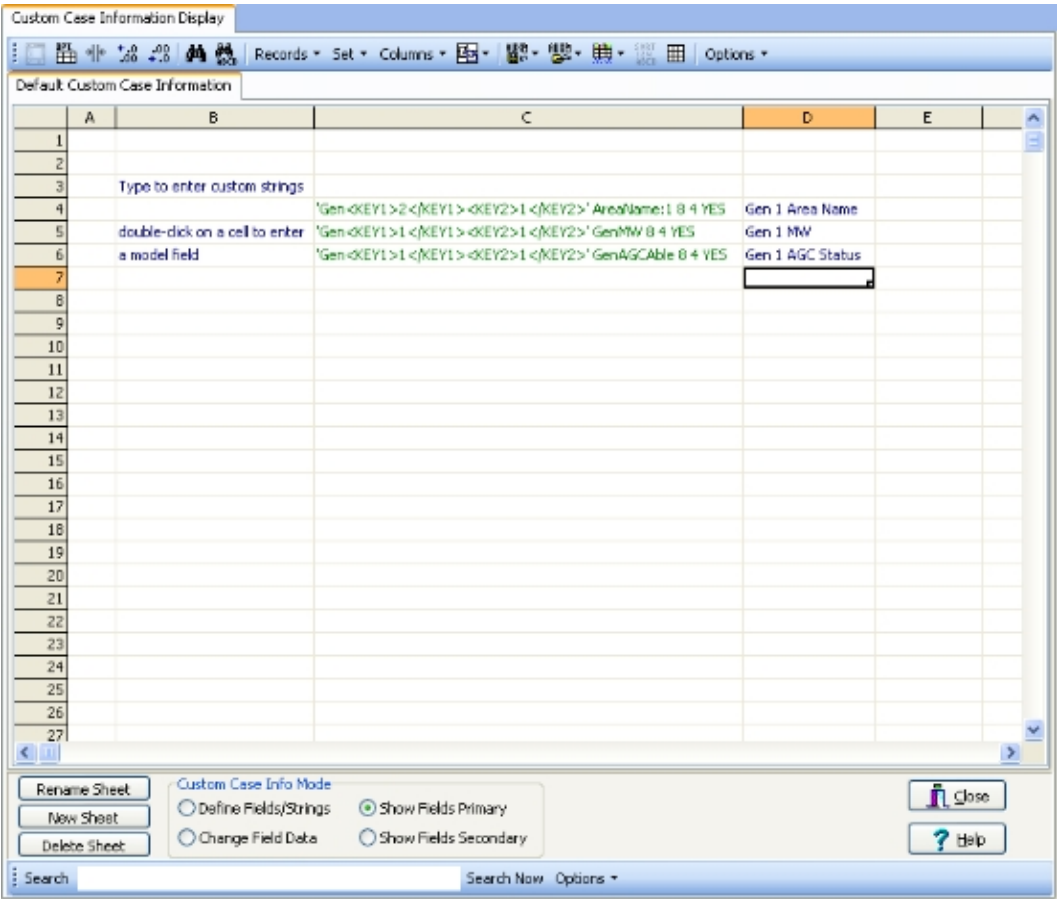
Model field cells will display a string which represents information about the model field link. The format for this string will be

``model field name' `variable name' totaldigits decimalpoints IncludeUnits`

The model field string will use the primary [key fields](#).

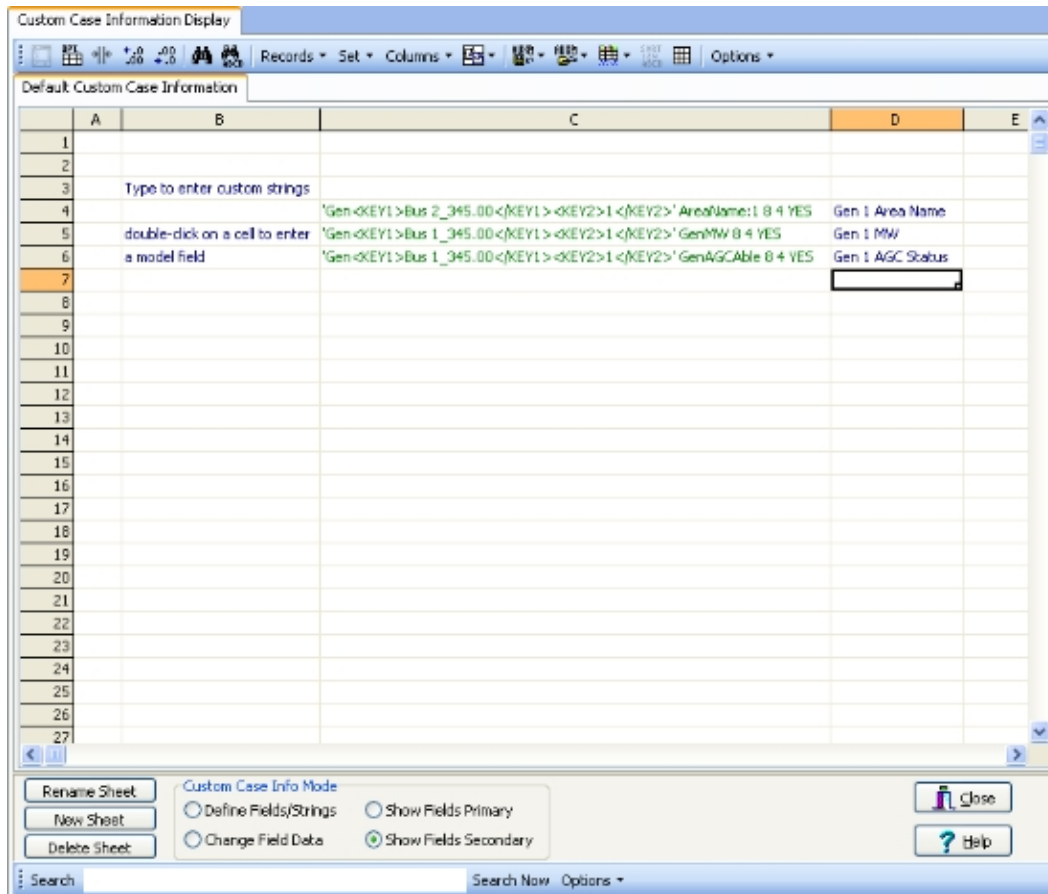
When pasting into such a cell or editing it directly it will parse the string entered trying to create a Model Field as though the string represents the model fields as shown in the Show Fields modes. If the string does not represent such a field, then it will remain a Plain Text Cell.





## Show Fields Secondary

This mode is identical to the [Show Fields](#) mode, except that it will show model fields string using the secondary [key fields](#).



---

**Custom Case Information Display Local Menu**


---

By right-clicking on the tabs representing the various sheets of the [custom case information display](#), a local menu appears giving you the following options.

- Rename Sheet – this is only available if you click directly on a specific tab. Choose it to rename the sheet.
- Delete Sheet – this is only available if you click directly on a specific tab. Choose it to delete the sheet.
- New Sheet – Choose this to add a new sheet
- Save All Sheets... – Choose this save all the information on the sheets to an auxiliary file. For the fields one the display, all information will be saved to the auxiliary file using [Primary Key Fields](#) unless the Custom Case Info Mode is set to Show Fields Secondary. In this case, they will be saved using the secondary key fields.
- Custom Case Info Mode – Choose this to open a submenu that allows you to change the mode.
- Tabs Position – Choose this to open a submenu that allows you to change the location of the tabs relative to the sheets
- Plain Text Change Data Color – Choose this to change the color which is used to denote Plain Text cells when the Custom Case Info Mode is set to [Change Field Data](#). The default color is dark gray.

By right-clicking on one of the Custom Case Information Display sheets, you will open a local menu that has many options which are the same as other [Case Information Displays](#). There are a few extra options listed as follows

- Insert Row – Choose this to insert a new row at the location of the presently selected cell
- Delete Row – Choose this to delete rows that are part of the present selection
- Insert Column – Choose this to insert a new column at the location
- Delete Column – Choose this to delete columns that are part of the present selection
- Delete Cell – Choose this to delete the presently selected cells
- Custom Case Info Menu – Choose this to open the same menu available by right clicking on the tabs representing the various sheets

---

## Case Description

---

The Case Description Dialog allows you to enter a text description of a case. The portion of the description that is saved with the case varies with case type:

PowerWorld Binary (\*.pwb) an unlimited number of lines are allowed

PTI Raw Data format (\*.raw) Two lines of text

GE EPC Data format (\*.epc) The case title and all the pre-title comments are read in as the case description

PowerWorld Case (\*.pwc) No case description supported

IEEE Common Format (\*.cf) No case description supported

In PowerWorld Viewer, these descriptions are read-only.

To display the Case Description Dialog to the [Case Information](#) ribbon tab, and choose **Case Description** from the **Case Data** ribbon group.

---

## Case Summary

---

The Case Summary Display provides a summary of the current case. Note that there are no enterable fields on the display. To display the Case Summary, go to the [Case Information](#) ribbon tab, and choose **Case Summary** from the **Case Data** ribbon group. The fields shown on this display include:

**Number of Devices in Case** shows the number of each of the following device types:

### Buses

Total number of buses in the case. Use the [Bus Display](#) to see a listing of these buses.

### Generators

Total number of generators in the case. Use the [Generator Display](#) to see a listing of these generators.

### Loads

Total number of loads in the case. Use the [Load Display](#) to see a listing of these loads.

### Switched Shunts

Total number of switched shunts in the case. Use the [Switched Shunt Display](#) to see a listing of these switched shunts.

### Trans. Lines (AC)

Total number of transmission lines in the case. Use the [Line/Transformer Display](#) to see a listing of these lines.

### LTCs (Control Volt)

Total number of Load Tap Changing transformers in the case. Use the [Transformer Control Display](#) to see a listing of these transformer controls.

### Phase Shifters

Total number of Phase Shifting transformers in the case. Use the [Transformer Control Display](#) to see a listing of these transformer controls.

### Mvar Controlling

Total number of Mvar Controlling transformers in the case. Use the [Transformer Control Display](#) to see a listing of these transformer controls.

### Series Capacitors

Total number of Series Capacitors in the case. These devices are treated the same as any other transmission line, and therefore can be found listed as a transmission element in the [Line/Transformer Display](#).

### 2 Term. DC Lines

Total number of two-terminal dc transmission lines in the case. Use the [DC Transmission Line Display](#) to see a listing of these dc lines.

### N-Term. DC Lines

Total number of multi-terminal dc transmission lines in the case. Use the [DC Transmission Line Display](#) to see a listing of these dc lines.

### Areas

Total number of areas in case. Use the [Area Display](#) to see a listing of these areas.

### **Zones**

Total number of zones in the case. Use the [Zone Display](#) to see a listing of these zones.

### **Islands**

Total number of islands in the case. An island is a group of buses that are interconnected through ac transmission lines or transformers but are isolated from the rest of the system. Each island must have a slack bus. In Simulator, use the Power Flow Solution tab of the [Simulator Options display](#) to specify whether multiple islands are allowed.

### **Interfaces**

Total number of interfaces in the case. An interface is a grouping of tie line objects between area objects. In Simulator, use the [Interfaces Display](#) to open the [Interface Dialog](#) to define and modify interface objects.

### **Injection Groups**

Total number of injection groups in the case. An injection group is a collection of loads and generators (objects that inject power into a network). In Simulator, use [Injection Groups Display](#) to open the [Injection Groups Dialog](#) to define and modify injection groups.

### **Case Totals**

Summarizes the total load, generation, shunt compensation, and losses for the case. Positive shunt compensation denotes shunt load, whereas negative shunt compensation indicates a shunt injection (such as shunt capacitance). The case totals fields are valid only when the current case is solved.

### **Generator Spinning Reserves**

The total difference between the present total generator output versus the total maximum possible output of all in-service generation.

### **Slack Buses:**

The slack bus or buses are listed showing the bus name, number, and area name and number. One slack bus is required for each island.

### **Case Pathname**

Full file name of the current case.

---

## Power Flow List

---

The Power Flow List shows detailed information about the system's power flows in a more traditional text-based form. This information is intended for users who would like detailed flow information about the power flow. Including the per unit voltage at the bus, the bus' load and generation, and flows on all lines and transformers emanating from the bus. The content of this display (i.e. which buses are included in the list) is governed by the [area/zone/owner filters](#).

To show this display go to the [Case Information](#) ribbon tab, and choose **Power Flow List** from the **Case Data** ribbon group.

To view flows at just a few select buses you may want to use the [Quick Power Flow List](#) instead. For large systems with no area/zone filtering set, it may take Simulator a long time to generate the complete Power Flow List. Note also that this display can show a maximum of 32,767 lines of text. If this limit is exceeded, Simulator will generate a resource error. Either use the [area/zone/owner filters](#) to limit the number of devices shown on this display, or use the Quick Power Flow List to focus on a few selected buses of interest.

The Power Flow List allows you to navigate through the system's buses rather easily. You can also use the display to show the flows for a bus' neighbor by double-clicking on the line that reads "TO nnnnn...", where nnnnn is the number of the bus you would like to see. The display is then positioned at this bus. If the bus is in an area and/or zone whose area/zone filter is not set, the area/zone filter is set automatically. In this way, you can inspect the system bus by bus.

The display can also be used to quickly display the dialog box associated with different bus objects. Place the cursor on the desired device, depress the CTRL key and then left click. The corresponding bus, load, generator, or shunt information dialogs will be displayed.

The Power Flow List also has its own local menu, which can be viewed by clicking the right mouse button on the display. Select **Change Font** to modify the style and size of the display's font. Select **Refresh** to ensure that the display's contents concur with the current system state. To skip to particular bus in the list, click **Find Bus**, which will open the [Find Bus Dialog](#). To display the information dialog for the currently selected branch, bus, load, or shunt, select **Display Object Dialog**. To print the display, choose the **Print** local menu option. Choosing **Copy** enables you to copy the display into the Windows clipboard, from where the information can be pasted into another application. Finally, select **Close** to close the display.

When printing the display you can either send the results directly to the printer or save them to a text file. To save the results in a text file on the Print Dialog select the **Save to File** option shown in the lower left corner of the dialog.

For each bus, the following items are shown:

### Bus

Shows the bus' number, name, and nominal voltage in kV. The next four fields are the MW, MVar, MVA and percentage headers for subsequent rows. The next fields specify the per unit voltage magnitude, voltage angle in degrees, the bus' area number and the bus' area name. Depress the CTRL key and then left click on this line to display the [Bus Dialog](#).

### Generator

For each generator at the bus, the Power Flow List shows the generator's ID (immediately after the keyword GENERATOR) and the power output of the generator in MW. Following this is generator's reactive power output in Mvar. A single character is shown immediately after the Mvar field. An 'R' indicates that the generator is regulating the bus voltage, 'H'

indicates that the generator is at its high reactive power limit, 'L' indicates that the generator is at its low reactive power limit, and a blank suggests that the generator is set off of AVR. The last field in the GENERATOR item is the MVA output of the generator. Depress the CTRL key and then left click on this line to display the [Generator Dialog](#). If no generators are connected to the bus, this item will be absent from the display.

#### **Load**

Shows the total power consumed by each load at the bus. If no loads are present at the bus, this item will be absent from the display. Depress the CTRL key and then left click on this line to display the [Load Dialog](#).

#### **Shunt**

Shows the total power for the fixed shunts at the bus. Positive shunt values denote shunt load, while negative shunt quantities indicate injection. If no shunts are connected to the bus, this item will be absent from the display.

#### **Switched Shunt**

Shows the total power for the switched shunts at the bus. Depress the CTRL key and then left click on this line to display the [Switched Shunt Dialog](#). If no switched shunts are located at the bus, this item will be absent from the display.

#### **Lines and Transformers**

For each line or transformer coming into the bus, the Power Flow List shows the line's flow and percentage loading. For transformers, the off-nominal tap ratio and phase shift angle in degrees are also shown. Immediately to the right of the off-nominal tap ratio is a two-character designation indicating the tapped side of the transformer: 'TA' indicates that the bus is on the tapped side, while 'NT' identifies the bus as residing on the side without the tap. You can left-click on this field to immediately reposition the bus to the other end of the line or transformer.

Depress the CTRL key and then left click on this line to display the [Transmission Line/Transformer Dialog](#). Simply left-click (without holding the CTRL key) to navigate through the system from bus to bus.



---

## Quick Power Flow List

---

The Quick Power Flow List provides a convenient means of viewing a listing of the flows at individual buses in the system. The format and control of the Quick Power Flow List is generally the same as that of the [Power Flow List](#), except that the Quick Power Flow List displays results for just the desired bus or [range of buses](#).

You can access this Quick Power Flow List in a number of different ways:

- Go to the [Case Information](#) ribbon tab, and choose **Quick Power Flow List** from the **Case Data** ribbon group
- From most of the Case Information Displays, right-click to invoke the display's local menu, and select *Quick Power Flow List*.
- From the [Bus Information Dialog](#) click on the *View All Flows at Bus* button.
- From the oneline diagram, right-click on the [bus symbol](#) to display the bus' local menu, and select *Quick Power Flow List*.

This display is automatically created if it is not already shown. Information on subsequent buses appears at the bottom of the display.

As with the Power Flow List, you can navigate through the system bus-by-bus by double-clicking on the lines that begin with "TO nnnnn ...," where *nnnnn* is the number of the bus you would like to investigate. Information for that bus will appear at the bottom of the display.

Like the Power Flow List, the Quick Power Flow List has a local menu that is accessed by right-clicking on the display. Among the things you can do from the local menu is to display the bus, branch, generator, load, or shunt corresponding to the currently selected record by selecting **Display Object Dialog**. You can also navigate through the system bus-by-bus just as you can do through double-clicking by choosing **Goto Line Bus** from the local menu.

---

## Outages

---

The outages display, available by choosing **Solution Details > Outages** in the [Model Explorer](#), presents a tabular listing of devices that are currently out-of-service in the load flow model. The display contains pages that resemble case information displays for [branches](#), [generators](#), [loads](#), [switched shunts](#), [buses](#), and [Multi-section lines](#). All devices listed in the pages on this display are the out-of-service elements.

---

## Area Display

---

The Area Display houses data about each area in the case. The Area Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [case information toolbar](#) from which you can print, copy, or modify its records as well as view the [information dialog](#) of its associated areas. You can also sort the area records by clicking on the heading of the field by which you want to sort.

To show the area records display, select **Aggregations > Areas** from the [Model Explorer](#) or **Aggregation > Areas** from the Case Information ribbon group under the [Case Information](#) ribbon tab.

By default, the area records display contains the following fields:

### Number, Name

Area's number, between 1 and 9999, and its alphanumeric identifier. Simulator supports names of any character length. However, when writing out names to file formats with limitations on name length, the area names will be truncated at their maximum supported length.

### AGC Status

Area's automatic generation control status. This field indicates whether or not the area's generation is changing automatically to control the area interchange. See [Area Control](#) for more details. You can toggle the value of the area's AGC Status (except in Viewer) by left clicking on the entry. Valid entries in this field include:

**Off AGC** Area is off AGC. Generation must be adjusted manually to meet changes in load and losses. If it is not, the system slack will be forced to pick up the balance.

**Part AGC** Area is on AGC, with generation dispatch controlled by its units' [participation factors](#).

**ED** Area is on economic dispatch control so that generation is dispatched in order of least cost.

**OPF** Area is on OPF control (only used with Simulator OPF). This option is only used with Simulator OPF. When the case is solved using the OPF the area controls are changed by the OPF to maintain area power balance. During non-OPF solutions this option is equivalent to Off AGC.

**Area Slack** Area is on Area Slack control. This option is only available for an area if you have already specified an area slack bus number in the area's [information dialog](#).

### Gen MW

Total real power generation in the area in MW.

### Load MW

Total real power load in the area in MW.

### Tot Sched MW

The net of the base and scheduled transactions between the area and all other areas, with exporting power indicated as a positive value. The interchange for an area is set on the [Area Information Dialog](#); the MW transactions for an area can be viewed using the specified [MW Transactions display](#).

### Int MW

The actual interchange between this area and all other areas, with exporting power positive. If an area is on AGC control, its actual interchange should match its scheduled interchange.

#### **ACE MW**

The area control error in MW. This is the amount of MW flow difference between the actual MW interchange and the desired MW interchange. A positive value means the super area is generating and exporting excess MW's, and a negative value means the super area is under-generating and importing too many MW's.

#### **Lambda**

The area's marginal cost. This marginal cost is relevant only when an Economic Dispatch (not OPF) solution has been run. Theoretically, if losses are ignored, an area operates most economically if all generators operate at the same incremental cost. This common incremental cost is the area's *lambda*, or marginal cost.

The local menu of the area records display has an additional option labeled *All Area Gen IC Curves*. Select this activity to generate a plot showing the incremental cost curves for all units located in a particular area.

#### **Loss (MW)**

Total real power losses for the area.

#### **Auto Shunts**

Determines whether switched shunts for the area are available for automatic control. You can use this field to disable all the switched shunts in an area. Click on this field to toggle its value. Click on the *Toggle All Yes* or *Toggle All No* local menu options to set the auto shunts property for all switched shunts. Note that a switched shunt is available for automatic control only if it meets three conditions: 1) its control mode property is set to *Automatic* (see [Switched Shunt Information Dialog](#)); 2) its associated area's *Auto Shunts* property is set to *Yes*; and 3) the *Disable Switched Shunt Control* option on the Power Flow Solution tab of the [Simulator Options Dialog](#) must not be checked.

#### **Auto XF**

Determines whether tapped transformers for the area are available for automatic control. You can use this field to disable all the transformers in an area. Click on this field to toggle its value. Click on the *Toggle All Yes* or *Toggle All No* local menu options to set all entries in this column. Note that three conditions must be met for a transformer to be used for automatic control: 1) its *Auto* field must be set to *Yes* (see [Transformer Modeling](#) for details); 2) its associated area's *Auto XF* property is set to *Yes*; and 3) the *Disable Transformer Control* option on the Power Flow Solution tab of the [Simulator Options dialog](#) must not be checked.

#### **Unspec. MW Inter.**

Shows the total amount of interchange for the area that is listed as Unspecified. This means that some of the interchange is known in magnitude and direction (import or export) but not which other area(s) is involved in the interchange. The sum of all area's unspecified interchange in the case must equal 0 for a balanced system.

---

## Zone Display

---

The Zone Display provides information about all the zones in the case. Similar to the [Area Display](#), the Zone Display provides a means of dividing up a power system. System results can then be summarized by zones using this display. Buses can be assigned to zones independent of their area assignments. Thus a single area could contain multiple zones, or a single zone could span multiple areas. The zone number for each bus is shown on the [Bus Dialog](#). In the Edit Mode, groups of buses can be easily moved from one zone to another using the [Zone Dialog](#).

The Zone Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated zones. You can call up the [Quick Power Flow List](#) or [Bus View Display](#) to obtain more information about a representative bus in the zone. You can also sort the zone records by clicking on the heading of the field by which you want to sort.

To show this display, click on **Aggregations > Zones** in the [model explorer](#).

The display contains the following fields by default:

### **Zone Number, Zone Name**

Zone's number, between 1 and 9999, and its alphanumeric identifier. Simulator supports names of any character length. However, when writing out names to file formats with limitations on name length, the zone names will be truncated at their maximum supported length.

### **Load MW, Load MVR**

Total real and reactive power load in the zone.

### **Gen (MW), Gen (Mvar)**

Total real and reactive power generation in the zone.

### **Loss MW, Loss MVR**

Total real and reactive power loss in the zone. Losses are computed by summing the losses of the individual transmission lines and transformers in the zone. Because of shunt charging, these devices can also generate reactive power. Therefore, reactive power losses may actually be negative.

### **Int MW, Int Mvar**

Net interchange of real and reactive power with all other zones. Exported power is assumed to be positive.

### **Load Mult MW**

Shows the current load MW multiplier for the zone. All load in the area is scaled by this multiplier.

---

## Tie Lines between Areas Display

---

The Tie Lines between Areas Display identifies all area tie lines in the case. Tie lines are not a separate data record that can be created in Simulator, but instead provide a look at all the devices that connect two areas together in the power system model. Tie lines are important because the [Area Control](#) algorithms used by Simulator often enforce a constraint which requires that the sum of the flow on the tie lines must equal the net export scheduled by the area (This is often called the ACE equation).

There are several types of devices that can represent tie lines. These include the following:

- Transmission branch (Branch) : a transmission line that connects two buses which belong to two different areas is a tie line.
- Two-terminal DC transmission lines (DCLine) : a DC transmission line that connects two buses which belong to two different areas is a tie line.
- Multi-terminal DC transmission line (MTDCRecord) : For every multi-terminal DC transmission line which has converters connected to more than one area is a tie line.
- Load (Load): A load can be assigned to a different area than its terminal bus. This is not typical but is sometimes done to model loads at lower voltage level where more than one entity has load that connects to the same power system bus. In such a situation this load represents a tie line.
- Generator (Gen): Similarly to the load, a generator can be assigned to a different area than its terminal bus. Again this is considered a tie line.
- Switched Shunt (Shunt): Similarly to the load, a switched shunt can be assigned to a different area than its terminal bus. Again this is considered a tie line.

Note: The easiest way to see which loads, generators, or switched shunts are assigned in this way is to use this Tie Lines between Areas Display. You can also however add separate columns using the display/column options on the Load, Generator or Switched Shunt Case Information Display. For instance for loads there area columns for Area Num of *Bus* and Area Num of *Load*. Normally these are equal, but they will not be for tie line loads.

Note: Be careful when working with cases opened as RAW files in which there are loads that represent tielines. See Add and Remove of transactions due to tie-line loads in [Simulator Options: File Management](#) for more information.

The Tie Lines between Areas Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated device type. You can learn more about a particular shunt's terminal bus by choosing either [Quick Power Flow List](#) or [Bus View](#). You can also sort the tie line information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Tie Lines Display according to the constraints of the [area/zone/owner filters](#).

To call up the Tie Lines between Areas Display, select **Aggregation > Tie Lines between Areas** from the [Model Explorer](#).

The Tie Lines between Areas Display contains the following fields by default:

### Tie Type

Indicates type of device which the tie line represents: Branch, Load, Gen, Shunt, MTDCRecord, or DCLine.

### Area Name, Number, Name

The Bus Number, Name and Area Name of the bus to which the device is attached.

**Far Area Name, Far Number, Far Name**

The Bus Number, Name and Area Name of the other end of the device.

**Ckt**

The circuit ID for transmission lines or DC Lines, the device ID for generators, loads, and switched shunts.

**Meter MW and Mvar**

The MW and Mvar flow from the point represented by Number/Name towards the far Number Name. For transmission lines and DC lines the metered point on the line is determined by the Metered End which is stored with the device. For instance on a transmission line, there is a check box for From End Metered which is available on the [Transmission Line/Transformer Option \(Edit Mode\)](#) dialog. Changing this setting with the device will affect the Meter values slightly.

**Status**

This is the Open/Closed status of the tie line device.

**Lim MVA**

For tie lines that represent transmission lines this is the limit of the device.

**MW and Mvar Loss**

For tie lines that represent transmission lines this is the MW and Mvar loss across the device.

---

## **Tie Lines between Zones Display**

---

The Tie Lines between Zones Display is essentially identical to the [Tie Lines between Areas Display](#). The only difference is that it displays devices that connect two zones together instead of areas. See the [Tie Lines between Areas Display](#) for more details.



---

## Super Area Display

---

The Super Area Display identifies any super areas that have been defined for the case. Super areas are groups of areas whose generators are dispatched as a coordinated group. Super areas can thus be useful for modeling the operation of independent system operators or power pools.

Super areas cannot be inserted into a case from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Instead, a super area can be defined when modifying or creating an area simply by typing the name of a new super area in the *Super Area* dropdown box on the [Area Information Dialog](#).

The Super Area Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated super areas. You can call up the [Quick Power Flow List](#) or [Bus View Display](#) to obtain more information about representative bus in the super area. You can also sort the super area records by clicking on the heading of the field by which you want to sort.

To show this display select **Aggregation > Super Areas** in the [Model Explorer](#).

The display contains the following fields by default:

### Super Area

The name of the super area.

### AGC Status

The Super Area may operate without automatic generation control (AGC Status = *Off AGC*), with participation factor control (AGC Status = *Part. AGC*), or according to an economic dispatch (AGC Status = *ED*). This is a toggleable field.

### Use Area PF

Indicates whether to use the areas participation factors when the super area is to operate with participation factor control (Part. AGC).

### Num Areas

Indicates the number of areas defined as being part of the super area. Areas are added to super areas using the Super Area dropdown box on the [Area Information Dialog](#).

### Gen MW

Total MW injection from all the generators in the super area.

### Load MW

Total MW load demanded in the super area.

### Tot Sched MW

Total scheduled MW interchange with other areas or super areas.

### ACE MW

The area control error in MW. This is the amount of MW flow difference between the actual MW interchange and the desired MW interchange. A positive value means the super area is generating and exporting excess MW's, and a negative value means the super area is under-generating and importing too many MW's.

### Lambda

The super area's marginal cost.

**Loss MW**

Indicates the real power losses incurred within the super area.

**ED Use PF**

Indicates whether the power flow engine will calculate loss penalty factors in computing the economic dispatch solution for the super area. If loss penalty factors are not calculated, then the economic dispatch is calculated assuming that the super area is lossless.

Otherwise, the economic dispatch solution incorporates losses. The penalty factors gauge the sensitivity of the area's losses to changing injection at specific generators. The option to calculate loss penalty factors is relevant only when the super area operates according to *Economic Dispatch Control*. Usually, if the system's cost curves are relatively flat, the inclusion of losses in the solution will not have much of an effect on the dispatch.

---

## Bus Display

---

The Bus Display presents data describing each bus in the case. The Bus Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [case information toolbar](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated bus. You can also sort the bus records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the Bus Records Display according to the constraints of the [area/zone/owner filters](#). Finally, when Simulator is in [Edit Mode](#), the case information toolbar allows you to add new buses to or remove existing buses from the system.

To show the bus records display, select **Network > Buses** from the [Model Explorer](#) or from the Case Information ribbon group under the [Case Information](#) ribbon tab. By default, the bus records display contains the following fields:

### Number and Name

Bus number between 1 and 2,147,483,647 (equals  $2^{31}$  minus 1), and its alphanumeric identifier.

### Area Name

Alphanumeric identifier of the bus' area.

### Nom kV

The nominal base voltage of the bus in kV.

### PU Volt

Bus' per unit voltage magnitude.

### Volt (kV)

Bus' actual voltage magnitude in kV. This is the per unit voltage magnitude multiplied by the bus' nominal voltage.

### Angle (Deg)

Bus' voltage angle in degrees.

### Load MW, Load Mvar

Total real and reactive load at the bus. If no loads are located at the bus, these fields are blank.

### Gen MW, Gen Mvar

Total real and reactive generation at the bus. If no generators are located at the bus, these fields are blank.

### Switched Shunt Mvar

Total switched shunt device reactive power injection at the bus.

### Act G Shunt MW, Act B Shunt Mvar

Total real and reactive fixed bus shunt injections.

### Area Num

The area number in which the bus is located.

### Zone Num

The zone number in which the bus is located.

---

## Remotely Regulated Bus Display

---

The Remotely Regulated Bus Display provides information about all buses that are remotely regulated by one or more generators, transformers, or switched shunts. The bus that a generator regulates is specified in the Edit Mode using the [Generator Dialog](#). Whenever a generator is regulating a bus that is not its terminal, it is considered to be remotely regulating that bus. The bus that is remotely regulated, along with the regulating generators, will appear on this display. The same logic is true for transformers and switched shunts which may regulate remote buses also.

The Remotely Regulated Bus Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view [the information dialog](#) of its associated buses. You can find a specific remotely regulated bus, and you can learn more information about a particular remotely regulated bus by choosing either [Quick Power Flow List](#) or [Bus View](#). You can also sort the remotely regulated bus information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Load Display according to the constraints of the [area/zone/owner filters](#).

To call up the Remotely Regulated Bus Display, open the [Model Explorer](#) and select **Remotely Regulated Buses** under the Solution Details folder.

This Remotely Regulated Bus Display contains the following fields by default:

### **Number, Name, Area Name**

Number, name, and area name for the bus that is being remotely regulated.

### **PU Volt**

Per unit voltage magnitude for the bus.

### **Set Volt**

Setpoint voltage for the bus. When a bus is being remotely regulated by a set of generators, the generators vary their reactive power output to maintain the voltage at the bus at the setpoint value. You can enter a new value for this field. Changing the setpoint voltage here changes the setpoint voltage for all the generators that are remotely regulating this bus.

### **Volt Diff**

Per unit difference between the actual voltage magnitude and the set point voltage magnitude.

### **AVR**

Combined automatic voltage regulation (AVR) status for all the generators remotely regulating this bus. If AVR is *No*, no generators regulate voltage; if AVR is *Yes*, all the available generators are regulating; if AVR is *Mixed*, some generators regulate voltage and some do not. Regulation of individual generators can be specified using the [Generator Display](#). You can toggle this field between "Yes" and "No" by clicking on it.

### **Total Mvar**

Total of the reactive power being supplied by all the generators remotely regulating the bus.

### **MVR Min, MVR Max**

Total of the minimum and maximum reactive power limits for all the generators remotely regulating the bus.

**Remote Regs (gen)**

Provides a comma-separated list of all the generators which regulate this bus.

**Rem Regs (XFMR)**

Provides a comma-separated list of all the transformers which regulate this bus.

**Rem Regs (SS)**

Provides a comma-separated list of all the switched shunts which regulate this bus.

---

## Bus Mismatches Display

---

The Bus Mismatches Display lists the real and reactive mismatches at each bus. The bus mismatches are defined as the difference between the power entering the bus and the power leaving the bus. A power flow case is considered solved when all the bus mismatches are below the convergence tolerance specified on the Power Flow Solution page of the [Simulator Options](#) dialog.

Most of the time you will not need to be concerned about the bus mismatches. If the power flow solves, the mismatches are guaranteed to be below the desired tolerance. However, advanced users will find this display useful in determining the cause when a power flow diverges.

The Bus Mismatch Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [case information toolbar](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated bus. You can find a specific bus mismatch using the name or number of the bus, and you can learn more about the bus by choosing either [Quick Power Flow List](#) or [Bus View](#). You can also sort the bus mismatch information by clicking on the heading of the field by which you want to sort.

To show this display select **Solution Details > Mismatches** from the [Model Explorer](#). You can view bus mismatches only when the application is in Run Mode.

Also note that there is a special feature in the Records menu of the case information toolbar for the mismatch display. This option is **Zero-Out Mismatches**. Choosing this options will cause Simulator to insert fictitious B and G Bus Shunt values to force the mismatch at every bus to zero.

The Bus Mismatch Display contains the following fields by default:

**Number, Name, Area Name**

The number and name of the bus and the name of the area in which it is located.

**MW Mismatch, MVR Mismatch, MVA Mismatch**

The real and reactive mismatches at each bus, and the total complex power mismatch.

---

## Substation Records Display

---

The Substation Records Display presents data describing each substation in the case. The Substation Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the information dialog of its associated substation. You can also sort the substation records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the Substation Records Display according to the constraints of the [area/zone/owner filters](#). Finally, when Simulator is in [Edit Mode](#), the local menu allows you to add new substations to or remove existing substations from the system. Note: a substation is considered in an area/zone if any single bus in the substation is in the area/zone.

To show the substation records display, select **Aggregation > Substations** in the [Model Explorer](#).

By default, the substation records display contains the following fields:

### **Sub Num**

An integer identifier for the substation.

### **Sub Name, Sub ID**

Two alphanumeric identifiers for the substation.

### **Area Name, Zone Name**

The names of the area and zone of the buses in the substation. If some of the buses in the substation are in different areas or zones, then this is the most common area or zone.

### **# of Buses**

The number of buses inside the substation.

### **Nom kV**

Nominal kV of the highest voltage bus(es) in the substation.

### **Gen MW, Gen MVR**

Total real and reactive generation at the substation. If no generators are located at the substation, these fields are left blank.

### **Load MW, Load MVR**

Total real and reactive load at the substation. If no loads are located at the substation, these fields are left blank.

### **Shunt MW, Shunt MVR**

Total real and reactive shunt values at the substation. If no shunts are located at the substation, these fields are left blank.

---

## Generator Display

---

The Generator Display presents data describing each generator in the case. The Generator Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated generators. The [Quick Power Flow List](#) and [Bus View Display](#) tools are available for finding more information on the generator's terminal bus. You can also sort the generator records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the Generator Records Display according to the constraints of the [area/zone/owner filters](#). Finally, you can use the local menu's *Insert* and *Delete* options when the application is in [Edit Mode](#) to insert a new generator into the case or to delete an existing generator.

Some columns which can be added to the generator records are related to the post-contingency solution options. For more information about these columns see the sections "Define Generator Line Drop and Reactive Current Compensation" and "Define Generator Maximum MW Responses in Post-Contingency" in the [Advanced Modeling Options](#) for [Contingency Analysis](#).

To show the generator records display, select **Network > Generators** from the [Model Explorer](#).

By default, the generator records display contains the following fields:

### Number, Name

Number and name of the bus to which the generator is attached. The display's local menu offers you the opportunity to view the [Quick Power Flow List](#) and the [Bus View Display](#) for the bus.

### ID

Single character ID used to distinguish multiple generators at the same bus. This default value for this field is '1'.

### Status

Displays the Open / Closed status of the generator. This field is a toggleable field.

### Gen MW, Gen Mvar

The real and reactive power output of the generator. If the generator is on AVR control, the reactive power is set automatically.

### Set Volt

Per unit setpoint voltage for the generator. When a generator is on AVR control, the reactive power output of the generator is varied automatically in order to maintain the regulated bus voltage at this value. The regulated bus is usually, but not always, the generator's terminal bus. Use the [Generator Dialog](#) to see the regulated bus number.

### AGC

Designates whether the generator's real power output is governed by automatic generation control. If the AGC field is set to *Yes*, the generator is on automatic generation control (AGC). When a generator is on AGC, its real power output can be varied automatically. Usually the purpose for AGC is to keep the area interchange at a desired value. You can click on this field to toggle its value (except in Viewer). Please see [Area Control](#) for more details.

### AVR



Designates whether the generator will vary its reactive power output to maintain a constant terminal voltage. If the *AVR* property is set to *Yes*, the generator is on automatic voltage regulation (AVR) control. When a generator is on AVR control, its reactive power output is varied automatically to keep the regulated bus voltage at the **Set Volt** value. AVR is limited by the generator's reactive power limits. You can click on this field to toggle its value (except in Viewer).

**Min MW, Max MW**

Minimum and maximum allowable real power output of the generator.

**Min Mvar, Max Mvar**

Minimum and maximum allowable reactive power output of the generator.

**Cost Model**

The type of cost model the generator is currently set to use. Cost models are necessary for performing economic analysis, such as Economic Dispatch or [Optimal Power Flow](#).

**Part. Factor**

Generator's participation factor. Participation factors are used to determine how AGC generators participate in area control when their area is on participation factor control. Please see [Area Control](#) for more details.

---

## **Generator/Load Cost Models**

---

The Generator/Load Cost Models option from the Case Information menu allows you to choose to view detailed information on generator cost curves or load benefit curves. This cost information can be very important when solving an [Optimal Power Flow](#) or [Security Constrained Optimal Power Flow](#) solution.

The Generator/Load Cost Models option has four submenu options to choose from:

[Generator Cubic Cost Models](#)

[Generator Piecewise Linear Cost Models](#)

[All Generator Cost Models](#)

[All Load Benefit Models](#)

---

## Generator Cost Models Display

---

The Generator Cost Models Display presents detailed cost information for each generator in the case, regardless of what type of cost curve has been entered for the generator. The Generator Cost Models Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated generators. The [Quick Power Flow List](#) and [Bus View Display](#) tools are available for finding more information on the generator's terminal bus. You can also sort the generator records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the Generator Cost Models Display according to the constraints of the [area/zone/owner filters](#). Finally, you can use the local menu's *Insert* and *Delete* options when the application is in [Edit Mode](#) to insert a new generator into the case or to delete an existing generator.

To show the generator piecewise linear and cubic cost model display, select **Network > Generators > Cost Curves All** from the [Model Explorer](#).

By default, the generator records display contains the following fields:

### Number, Name

Number and name of bus to which the generator is attached.

### Area Name of Gen

Name of the area to which the generator belongs. The generator can be assigned to an area different than the area to which its terminal bus is assigned.

### ID

Single character ID used to distinguish multiple generators at the same bus; '1' by default.

### Status

The Open / Closed status of the generator. This is a toggleable field.

### AGC

Designates whether the generator's real power output is governed by automatic generation control. If the AGC field is set to *Yes*, the generator is on automatic generation control (AGC). When a generator is on AGC, its real power output can be varied automatically. Usually the purpose for AGC is to keep the area interchange at a desired value. You can click on this field to toggle its value (except in Viewer). Please see [Area Control](#) for more details.

### Gen MW

Current real power output of the generator.

### Min MW

Minimum MW output of the generator.

### Max MW

Maximum MW output of the generator.

### Cost Model

The type of model this generator is currently using. Can be Cubic, Piecewise Linear or None.

### IOA, IOB, IOC, IOD

Parameters used to model the cost characteristic of the generator. Please see [Generator Cost Information](#) for details. Please note that these values can be saved/loaded using the [Generator Cost Data](#) auxiliary file.

These fields will be disabled unless the Cost Model type is set to Cubic.

#### **Fuel Cost**

The fuel cost of the type of fuel for the generator.

#### **Variable O&M**

Operations and Maintenance costs for the generator.

This field will be disabled unless the Cost Model type is set to Cubic.

#### **Fuel Type**

An informational field that can be set to the type of fuel the generator uses.

#### **Unit Type**

An informational field that can be set to reflect the type of unit the generator is, such as combined cycle, steam, hydro, etc.

#### **Cost Shift \$/MWh, Cost Multiplier**

The cost shift and cost multiplier allow you to easily apply a shift to the cost function for the purpose of assessing how variations in bids impact profit. The cost function is affected based on the following equation:

$(\text{Original Cost Function} + \text{Cost Shift}) * \text{Cost Multiplier}$

#### **Fixed Cost**

The fixed operating cost of the generator.

#### **MWh Break x, MWh Price x**

The remainder of the display is populated with MWh Break and MWh Price pairs. These pairs define the break points of the piecewise linear curve. The MWh Break value is a MW output value of the generator. The MWh Price value is the corresponding marginal cost of producing an additional MW of power at that MW output level. Therefore entering the break points of the piecewise linear curve in this manner defines the slopes of the next section of the curve, starting at the current MW Break point and up to but not including the next defined break point. The last MWh Break and MWh Price pair defined will define the marginal price of the unit from that break point location to the maximum output of the generator.

A requirement of the piecewise linear cost curve is that it must be convex, meaning the next MWh Price must be higher than the previous MWh Price.

These fields will be disabled unless the Cost Model type is set to Piecewise Linear.

---

## Generator Cubic Cost Display

---

The Generator Cubic Cost Display presents detailed cost information for each generator in the case set to use a cubic cost model. The Generator Cubic Cost Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated generators. The [Quick Power Flow List](#) and [Bus View Display](#) tools are available for finding more information on the generator's terminal bus. You can also sort the generator records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the Generator Cubic Cost Display according to the constraints of the [area/zone/owner filters](#). Finally, you can use the local menu's *Insert* and *Delete* options when the application is in [Edit Mode](#) to insert a new generator into the case or to delete an existing generator.

To show the generator cubic cost display, select **Network > Generators > Cost Curves Cubic** from the [Model Explorer](#).

By default, the generator records display contains the following fields:

### Number, Name

Number and name of bus to which the generator is attached.

### Area Name of Gen

Name of the area to which the generator belongs. The generator can belong to an area which is different than the area of which its terminal bus is a member.

### ID

Alphanumeric ID used to distinguish multiple generators at the same bus; '1' by default.

### Status

The Open / Closed status of the generator. This is a toggleable field.

### AGC

Designates whether the generator's real power output is governed by automatic generation control. If the AGC field is set to *Yes*, the generator is on automatic generation control (AGC). When a generator is on AGC, its real power output can be varied automatically. Usually the purpose for AGC is to keep the area interchange at a desired value. You can click on this field to toggle its value (except in Viewer). Please see [Area Control](#) for more details.

### Gen MW

Current real power output of the generator.

### IOA, IOB, IOC, IOD

Parameters used to model the cost characteristic of the generator. Please see [Generator Cost Information](#) for details. Please note that these values can be saved/loaded using the [Generator Cost Data](#) auxiliary file.

### Fuel Cost

The fuel cost of the type of fuel for the generator.

### Variable O&M

Operations and Maintenance costs for the generator.

### Fuel Type

An informational field that can be set to the type of fuel the generator uses.

#### **Unit Type**

An informational field that can be set to reflect the type of unit the generator is, such as combined cycle, steam, hydro, etc.

#### **Cost Shift \$/MWh, Cost Multiplier**

The cost shift and cost multiplier allow you to easily apply a shift to the cost function for the purpose of assessing how variations in bids impact profit. The cost function is affected based on the following equation:

$$(\text{Original Cost Function} + \text{Cost Shift}) * \text{Cost Multiplier}$$

#### **Cost \$/Hr**

Operating cost for the generator in \$/hr.

#### **IC**

Incremental cost to produce an additional MWh. This can be expressed as  $dC(P_{gi})/dP_{gi}$ , where C denotes the generator's cost of operation in \$/hr and  $P_{gi}$  expresses the current MW output of the unit. In a lossless system, the incremental dispatch is equal to the generator's lambda value.

#### **LossSens**

Area loss sensitivity - This field is **only calculated when the generator's area is on economic dispatch control**. This field specifies the incremental change in area losses if this generator were to produce one more MW, **with the excess generation absorbed by the system slack**. This may be expressed as  $\partial P_{loss}/\partial P_{gi}$ . The loss sensitivity is used in calculating the generator's lambda value for the economic dispatch activity.

#### **Generator MW Marg. Cost**

The marginal cost of the generator supplying an additional MW of power to the system.

---

## Generator Piecewise Linear Cost Display

---

The Generator Piecewise Linear Cost Display presents detailed cost information for each generator in the case set to use a piecewise linear cost model. The Generator Piecewise Linear Cost Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated generators. The [Quick Power Flow List](#) and [Bus View Display](#) tools are available for finding more information on the generator's terminal bus. You can also sort the generator records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the Generator Piecewise Linear Cost Display according to the constraints of the [area/zone/owner filters](#). Finally, you can use the local menu's *Insert* and *Delete* options when the application is in [Edit Mode](#) to insert a new generator into the case or to delete an existing generator.

To show the generator piecewise linear cost display, select **Network > Generators > Cost Curves Linear** in the [Model Explorer](#).

By default, the generator records display contains the following fields:

### Number, Name

Number and name of bus to which the generator is attached.

### Area Name of Gen

Name of the area to which the generator belongs. The generator can belong to an area which is different than the area of which its terminal bus is a member.

### ID

Alphanumeric ID used to distinguish multiple generators at the same bus; '1' by default.

### Status

The Open / Closed status of the generator. This is a toggleable field.

### AGC

Designates whether the generator's real power output is governed by automatic generation control. If the AGC field is set to *Yes*, the generator is on automatic generation control (AGC). When a generator is on AGC, its real power output can be varied automatically. Usually the purpose for AGC is to keep the area interchange at a desired value. You can click on this field to toggle its value (except in Viewer). Please see [Area Control](#) for more details.

### Gen MW

Current real power output of the generator.

### Min MW

Minimum MW output of the generator.

### Max MW

Maximum MW output of the generator.

### Fuel Cost

The fuel cost of the type of fuel for the generator.

### Fuel Type

An informational field that can be set to the type of fuel the generator uses.

**Unit Type**

An informational field that can be set to reflect the type of unit the generator is, such as combined cycle, steam, hydro, etc.

**Cost Shift \$/MWh, Cost Multiplier**

The cost shift and cost multiplier allow you to easily apply a shift to the cost function for the purpose of assessing how variations in bids impact profit. The cost function is affected based on the following equation:

$$(\text{Original Cost Function} + \text{Cost Shift}) * \text{Cost Multiplier}$$

**Fixed Cost**

The fixed operating cost of the generator.

**MWh Break x, MWh Price x**

The remainder of the display is populated with MWh Break and MWh Price pairs. These pairs define the break points of the piecewise linear curve. The MWh Break value is a MW output value of the generator. The MWh Price value is the corresponding marginal cost of producing an additional MW of power at that MW output level. Therefore entering the break points of the piecewise linear curve in this manner defines the slopes of the next section of the curve, starting at the current MW Break point and up to but not including the next defined break point. The last MWh Break and MWh Price pair defined will define the marginal price of the unit from that break point location to the maximum output of the generator.



---

## Load Display

---

The Load Display presents data describing each load in the case. The Load Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [case information toolbar](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated loads. The toolbar also affords the opportunity to insert new loads into the model or to delete existing ones. Moreover, it enables you to invoke the [Quick Power Flow List](#) and [Bus View Display](#) for the load's terminal bus. You can also sort the load information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Load Display according to the constraints of the [area/zone/owner filters](#).

To show the load display, select **Network > Loads** in the [Model Explorer](#).

The load display contains the following fields by default:

### **Number, Name**

Number and name of bus to which the load is attached.

### **ID**

Two-character ID used to distinguish multiple loads at the same bus; '1' by default.

### **Status**

Either Closed if the load is connect to its bus, or Open if it is not. You can click on this field to toggle its value. If the load is open, the entire load record is dimmed.

### **MW, Mvar, MVA**

Total real, reactive, and complex power for the load. Loads may be both voltage dependent. The total load is the sum of the constant power, constant current, and constant impedance components. See [Load Information](#) and see [Load Modeling](#) for more information.

### **S MW, S MVAR, I MW, I MVR, Z MW, Z MVR**

These six fields describe the composition of the load at the bus assuming 1 pu bus voltage. The SMW and SMVAR fields indicate the constant power portion of the load, the component that does not vary with bus voltage magnitude. The IMW and IMVR fields express the constant current part of the load, which varies in proportion to the bus voltage magnitude. Finally, ZMW and ZMVR indicate the constant impedance portion of the load, which varies with the square of the voltage. The sum of the SMW, IMW, and ZMW fields yields the base MW load at the bus (assuming 1pu voltage), and the sum of the SMVR, IMVR, and ZMVR fields provides the base MVR load at the bus (assuming 1 pu voltage). Please see [Load Modeling](#) for more details on how bus load is modeled.

---

**Load Benefit Models Display**


---

The Load Benefit Models Display presents detailed cost information for each load in the case set to use a piecewise linear benefit model. The Load Benefit Models Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated loads. The [Quick Power Flow List](#) and [Bus View Display](#) tools are available for finding more information on the load's terminal bus. You can also sort the load records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the Load Benefit Models Display according to the constraints of the [area/zone/owner filters](#). Finally, you can use the local menu's *Insert* and *Delete* options when the application is in [Edit Mode](#) to insert a new load into the case or to delete an existing load.

To show the generator piecewise linear cost display, select **Network > Loads > Benefit Curves Linear** in the [Model Explorer](#).

By default, the generator records display contains the following fields:

**Number, Name**

Number and name of bus to which the load is attached.

**Area Name of Gen**

Name of the area to which the load belongs. The load can belong to an area which is different than the area of which its terminal bus is a member.

**ID**

Alphanumeric ID used to distinguish multiple generators at the same bus; '1' by default.

**Status**

The Open / Closed status of the load. This is a toggleable field.

**AGC**

Designates whether the generator's real power output is governed by automatic generation control, since effectively dispatching a load can be viewed as dispatching negative generation. If the AGC field is set to *Yes*, the load is on automatic generation control (AGC). When a load is on AGC, its real power output can be varied automatically. Usually the purpose for AGC is to keep the area interchange at a desired value. You can click on this field to toggle its value (except in Viewer). Please see [Area Control](#) for more details.

**Gen MW**

Current real power demand of the load.

**Min MW**

Minimum MW demand of the load.

**Max MW**

Maximum MW demand of the load.

**Fixed Benefit**

The fixed benefit of the load.

**Benefit Model**

The type of model this load is currently using. Can be either Piecewise Linear or None.

**MWh Break x, MWh Price x**

The remainder of the display is populated with MWh Break and MWh Price pairs. These pairs define the break points of the piecewise linear curve. The MWh Break value is a MW demand value of the load. The MWh Price value is the corresponding marginal benefit of extracting an additional MW of load at that MW output level. Therefore entering the break points of the piecewise linear curve in this manner defines the slopes of the next section of the curve, starting at the current MW Break point and up to but not including the next defined break point. The last MWh Break and MWh Price pair defined will define the marginal benefit of the load from that break point location to the maximum demand of the load.

A requirement of the piecewise linear benefit curve is that it must be concave, meaning the next MWh Price must be lower than the previous MWh Price. In other words, as more load is supplied, the less the benefit it is providing. This is how a load can be dispatched along with generation, according to marginal costs (and marginal benefits.)

These fields will be disabled unless the Cost Model type is set to Piecewise Linear.

---

## Line and Transformer Display

---

The Line/Transformer Display presents data describing each transmission line and transformer in the case. The Line/Transformer Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated branches. The local menu also affords the opportunity to insert new lines or transformers into the model or to delete existing ones. Moreover, it enables you to invoke the [Quick Power Flow List](#) and [Bus View Display](#) for each branch's terminal buses. You can also sort the line and transformer information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Line/Transformer Display according to the constraints of the [area/zone/owner filters](#).

To show the line/transformer display, select **Network > Branches Input** or **Network > Branches State** from the [Model Explorer](#).

The contents of the display depends upon the application's operating mode.

The line/transformer display shows the following fields by default:

### From Bus Number and Name

*From* Bus number and name. For transformers, the *from bus* is the tapped side. Right-clicking on either of these fields brings up the display's local menu from which you may select [Quick Power Flow List](#) or [Bus View Display](#) to obtain more information about the *from bus*.

### To Bus Number and Name

*To* Bus number and name. Right-clicking on either of these fields brings up the display's local menu from which you may select [Quick Power Flow List](#) or [Bus View Display](#) to obtain more information about the *to bus*.

### Circuit

Two-character identifier used to distinguish between multiple lines joining the same two buses. Default is '1'.

### Status

The service status of the branch. This field is toggleable.

### Xfrmr

Yes or no field signifying if the branch is a transformer or transmission line. This field cannot be changed.

### From MW, From Mvar, From MVA (State display default)

Real, reactive, and complex power flowing into the line at the *from bus*.

### Lim MVA, Max Percent (State display default)

The current MVA limit of the branch, and the amount of the actual flow as a percentage of the MVA limit.

### MW Loss, Mvar Loss (State display default)

Real and reactive power losses on the transmission line or transformer. Since reactive power losses include the effect of line charging, the reactive power losses may be negative.

### R, X, B (Input display default)

Indicates the branch's resistance, reactance, line charging susceptance in per unit on the system base.

**Lim A MVA, Lim B MVA, Lim C MVA (Input display default)**

Identifies the first three limit settings. Five additional limits can be set, for which the columns can be added using the Display/Column options from the local popup menu. All limits are expressed in MVA.

See [Transformer Display](#) for viewing transformer specific fields.

---

## Multi-Section Lines Display

---

The Multi-Section Line Display lists the multi-section lines that exist in the case. The Multi-Section Line Display is used to functionally group a number of transmission lines together. They are usually used to model very long transmission lines that require the use of multiple individual transmission line information to be modeled accurately. Simulator then treats each multi-section line as a single device with regard to line status. That is, changing the status of one line in the record changes the status of the other lines in the record as well.

Each multi-section line record consists of the "from" bus, one or more "dummy" buses and the "to" bus. Each dummy bus must have only two lines connected to it, each of which are members of the multi-section line record. See [Multi-Section Line Information](#) for details.

The Multi-Section Line Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its multi-section line information. The local menu also affords the opportunity to insert new multi-section lines into the model or to delete existing ones when the application operates in the Edit Mode. Moreover, it enables you to invoke the [Quick Power Flow List](#) and [Bus View Display](#) for the line's terminal buses. You can also sort the multi-section line information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Multi-Section Line Display according to the constraints of the [area/zone/owner filters](#).

To show this display select **Aggregations > Multi-Section Lines** in the [Model Explorer](#).

The display contains the following fields by default:

### From Bus #, From Name

Number and name of the multi-section lines *from bus*. Right clicking on one of these fields invokes the display's local menu from which you can select either [Quick Power Flow List](#) or [Bus View Display](#) to find more information about the *from bus*.

### To Bus #, To Name

Number and name of the multi-section lines *to bus*. Right clicking on one of these fields invokes the display's local menu from which you can select either [Quick Power Flow List](#) or [Bus View Display](#) to find more information about the *to bus*.

### Circuit

Two-character circuit identifier for the multi-section line. The first character in the identifier should always be an "&."

### Sections

Number of individual lines within the multi-section line record.

### Allow Mixed Status

This value defaults to NO and means that all branches in the multi-section line must have the same status. Setting this value to YES means essentially means that the multi-section line is disabled so that opening/closing a branch in the multi-section line will no longer automatically open or close the other lines in the record.

### Status

Current status of the record. Note, the status is *Closed* only if **all** the lines in the record are closed, and is *Open* only if **all** the lines in the record are open. Otherwise the status is *Mixed*.



---

## Transformer Display

---

The Transformer Display identifies all transformers in the case. The data presented in the Transformer Display supplements the data presented in the [Line/Transformer Display](#) by presenting transformer-specific information. Consult the Line/Transformer Display for the transformer flows.

The Transformer Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated transformer. You can find a specific transformer using the names or numbers of its terminal buses, and you can learn more about a particular transformer's terminal buses by choosing either [Quick Power Flow List](#) or [Bus View](#). When in Edit Mode, you can delete an existing transformer from the case. You can also sort the transformer information by clicking on the heading of the field by which you want to sort.

To show this display select **Network > Transformers Controls** in the [Model Explorer](#).

The Transformer Display contains the following fields by default:

### From Bus Number and Name

*From Bus* number and name. The *From Bus* is the tapped side. You may view either the [Quick Power Flow List](#) or the [bus view display](#) for the *From Bus* from the local menu.

### To Bus Number and Name

*To Bus* number and name. You may view either the [Quick Power Flow List](#) or the [bus view display](#) for the *To Bus* from the local menu.

### Circuit

Two-character identifier used to distinguish between multiple transformers joining the same two buses.

### Status

The service status of the transformer. This field is toggleable.

### Type

Type of transformer. Possible values include

**Fixed** The tap positions are fixed

**LTC** The tap ratio changes to regulate bus voltage

**Mvar** The tap ratio changes to regulate reactive power flow

**Phase** The phase angle changes to regulate real power flow

### Tap/Phase

Indicates the tap ratio for LTC and fixed transformers and the phase shift angle in degrees for phase-shifting transformers.

### XF Auto

If the value of this field is *Yes*, the transformer will automatically change its tap or phase angle to keep the regulated value within the specified regulation range, provided that the *Auto XF* field of its associated area is set to *Yes* and transformer tap/phase control has not been disabled for the entire case. The *Auto XF* field of individual areas is set from the [Area Display](#), and case-wide transformer control can be set from the Power Flow Solution tab of the [Simulator Options](#) Dialog. Click on this field to toggle its values.



**Reg Bus**

For an LTC transformer, this is the number of the bus whose voltage is controlled by the transformer. For phase shifting transformers, the real power is always controlled at the tapped bus.

**Reg Value**

For an LTC transformer, this is the present per unit voltage at the regulation bus. For a phase shifting transformer, this is the present real power flow through the transformer measured on the "from" (tapped) side.

**Reg Error**

The error is the difference between the regulated value and the respective limit of the regulation range specified by Reg Min and Reg Max. If the regulated value is within the regulation range, then the error is zero. The error is negative if the regulated value falls below the regulation range, and it is positive if the regulated value exceeds the regulation range.

**Reg Min, Reg Max**

Minimum and maximum values for the regulation range. For LTC transformers, these fields represent per unit voltage at the regulated bus. For phase shifting transformers, these fields represent actual MW flow through the transformer measured on the "from" (tapped) side. Because transformers use discrete control, the maximum regulation value must be somewhat greater than the minimum value.

**Tap Min, Tap Max**

For LTC transformers, these fields specify the minimum and maximum tap ranges for the transformer. For phase shifting transformers, these fields specify the minimum and maximum phase shift angle in degrees.

**Step Size**

The per unit step size for tap changing transformers. This step size is usually determined by dividing the total range of transformer operation by the number of tap positions available.

---

### Three Winding Transformer Display

---

The Three Winding Transformer Display shows information about all the three winding transformer devices in the case. Three winding transformers are modeled in Simulator as a grouping of two winding transformers, connected at a common midpoint or "Star" bus. The Three Winding Transformer Display is a way to view all the terminal connection points for the three winding transformers and the power delivered at each of the terminals.

The Three Winding Transformer Display is a class of [Case Information Display](#), and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated three winding transformers. When in Edit Mode, you can define new three winding transformers using the **Insert** option, or delete existing three winding transformers using **Delete**. You can also sort the three winding transformer information by clicking on the heading of the field by which you want to sort.

To show this display, open the [model explorer](#) and click on **Network > Three Winding Transformers**.

This display contains the following fields by default:

**Pri Bus Num, Sec Bus Num, Ter Bus Num**

These are the primary winding, secondary winding and tertiary winding terminal bus numbers for the three winding transformer connections.

**Circuit**

The circuit identifier for the three winding transformer.

**Pri, Sec and Ter MW and MVar**

These fields display the real and reactive power delivered at each of the three winding transformer terminals.

**Star Bus Number and Name**

These two fields provide the number and name of the internal node star bus used in the three winding transformer equivalent model.

---

## Line Shunts Display

---

The Line Shunts Display identifies all line shunt devices in the case. Line shunts are similar to [switched shunts](#) in that they are used in power systems either to inject additional MVR into the system (capacitive shunts) or to absorb excess reactive power (inductive shunts). However instead of being connected to a bus directly, they are instead shunt connected to one of the terminals of a transmission line. As a result they can only be in-service if the actual transmission line is in-service. Also, Simulator also does not allow the automatic regulation of a bus voltage using Line Shunts as is possible using a switched shunt. Simulator does however allow line shunts to be individually taken in and out of service. This must be done manually however using the status field of the line shunt.

The Line Shunts Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated line shunts. You can find a specific line shunt using its bus name or number, and you can learn more about a particular shunt's terminal bus by choosing either [Quick Power Flow List](#) or [Bus View](#). When in Edit Mode, you can insert a new line shunt into the case or delete an existing shunt. You can also sort the line shunt information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Line Shunts Display according to the constraints of the [area/zone/owner filters](#).

To call up the Line Shunts Display, click **Network > Line Shunts** in the [Model Explorer](#).

The Line Shunts Display contains the following fields by default:

### From Number and To Number

Numbers of from and to bus of the transmission branch to which the line shunt is attached. Use the right menu options to inspect either the [Quick Power Flow List](#) or the [Bus View Display](#) for the terminal buses.

### Circuit

Circuit identifier for the transmission branch to which the line shunt is attached.

### Bus Number Located At

Either the From or To Bus number. Specified the end of the transmission branch to which the line shunt is attached.

### ID

A circuit identifier for the actual line shunt. This is needed because there can be multiple line shunts at the same end of the same transmission line.

### G, B

Indicates the line shunts nominal conductance and susceptance in per unit on the system base.

### Status

Line shunts can be individually taken in and out of service. Do this by changing the status field to Open or Closed.

---

## DC Lines Display

---

The DC Line Display presents data describing each dc line in the case. The DC Line Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its records as well as view the [information dialog](#) of its associated dc lines. You can also sort the dc line records by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the records shown by the DC Line Records Display according to the constraints of the [area/zone/owner filters](#).

To show the dc line records display, select **Network > DC Transmission Lines** in the [Model Explorer](#), and click on the DC Lines tab.

By default, the dc line records display contains the following fields:

### Number

Each dc line must be assigned a unique number, typically between 1 and 40.

### Rect AC Number, Rect AC Name, Rect MW, Rect Mvar

Number and name of the rectifier bus, and the real and reactive power flow from the rectifier into the dc line. You may right-click on any of the rectifier-related fields and select [Quick Power Flow List](#) or [Bus View](#) from the local menu to view additional information about the rectifier bus.

### Inv AC Number, Inv AC Name, Inv MW, Inv Mvar

Number and name of the inverter bus, and the real and reactive power flow from the inverter into the dc line. You may right-click on any of the inverter-related fields and select [Quick Power Flow List](#) or [Bus View](#) from the local menu to view additional information about the inverter bus.

### Control Mode

Specifies how flow on the dc line is controlled. If this field is set to *Power*, then the line's MW flow is the control parameter. If the control mode is defined as *Current*, then the line's current is the control parameter.

### Setpoint Magnitude, Setpoint Location

Specifies the initial value of the control parameter, which is expressed either in MW or in amps depending on the dc line's control mode. The Setpoint Location indicates if the setpoint should be set at the Inverter or Rectifier end of the DC line.

### Set kV

Specifies the voltage of the dc line in kV.

---

## Multi-Terminal DC Record Display

---

The Multi-terminal DC Line Display presents data describing each multi-terminal DC line in the case. The Multi-Terminal DC Line Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated records. The local menu also affords the opportunity to insert multi-terminal DC line records into the model or to delete existing ones.

To show the Multi-Terminal DC Line display, select **Network > DC Lines** in the [Model Explorer](#), and click on the Multi-terminal DC Lines tab of the transformer case information display.

The Multi-terminal DC Line display shows the following fields by default:

### **Number**

The record number of each multi-terminal DC record.

### **Num Conv**

The total number of converters in each multi-terminal DC network.

### **Num Buses**

The total number of DC buses in each multi-terminal DC network.

### **Num Lines**

The total number of DC lines in each multi-terminal DC network.

### **Mode**

The control mode of the multi-terminal DC network: 0 is Blocked control, 1 is Power control, and 2 is Current control.

### **V. Cont. Bus**

The number of the AC converter bus at which the DC voltage is controlled.

---

## Switched Shunt Display

---

The Switched Shunt Display identifies all [switched shunt](#) devices in the case. Switched shunts are used in power systems either to inject additional MVR into the system (capacitive shunts) or to absorb excess reactive power (inductive shunts). They may also be used to regulate bus voltage within some specified range.

The Switched Shunt Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated switched shunts. You can find a specific switched shunt using its bus name or number, and you can learn more about a particular shunt's terminal bus by choosing either [Quick Power Flow List](#) or [Bus View](#). When in Edit Mode, you can insert a new shunt into the case or delete an existing shunt. You can also sort the switched shunt information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Load Display according to the constraints of the [area/zone/owner filters](#).

To call up the Switched Shunt Display, select **Network > Switched Shunts** in the [Model Explorer](#).

The Switched Shunt Display contains the following fields by default:

### Number, Name

Number and name of terminal bus to which the switched shunt is attached. Not that multiple switched shunts are allowed at a bus, however only one switched shunt can have a Control Mode which is not fixed. Use the right menu options to inspect either the [Quick Power Flow List](#) or the Bus [View Display](#) for the terminal bus.

### ID

Circuit identifier for the switched shunt device.

### Reg Bus Num.

Number of the bus whose voltage is regulated by the switched shunt. If the regulation bus is the same as the terminal bus, this field is blank.

### Status

The service status of the switched shunt. This field is toggleable.

### Control Mode

Control Mode for the switched shunt. A switched shunt may operate either as *Fixed* if its reactive value is to be held constant, as *Discrete* if its reactive output is controlled automatically in discrete steps to regulate its terminal voltage, or as *Continuous* if its reactive injection is allowed to vary over a continuous spectrum of values to regulate its terminal voltage. A shunt will be switched either discretely or continuously to regulate its terminal bus' voltage between its high and low voltage limits provided the *Auto Shunt* field of its corresponding area is set to *Yes* and switched shunt control has not been disabled for the entire case. You may set the value of the *Auto Shunt* field from the [Area Display](#), and you can control case-wide enforcement of shunt switching from the Power Flow Solution tab of the [Simulator Options](#) Dialog. This Control Mode field is toggleable.

### Regulates

Can be set to regulate either bus voltage or generator MVAR output.

### Actual Mvar

The reactive power currently supplied by the switched shunt.

**Volt High**

Per unit high-voltage limit for the regulation range. It is important for discrete shunt control that Volt High exceed Volt Low by a nontrivial amount; otherwise, the output of the shunt may oscillate during the Power Flow Solution.

**Volt Low**

Per unit low-voltage limit for the regulation range.

**Reg Volt**

Actual per unit voltage at the regulated bus. When Control Mode is either *Discrete* or *Continuous*, this voltage should be between Volt Low and Volt High.

**Deviation**

Deviation of the regulated bus' actual per unit voltage from the desired regulation voltage. If the actual voltage is within the regulation range, this field is zero. If the voltage is greater than Volt High, the deviation is positive. It is negative if the actual voltage is less than Volt Low.

**Nominal Mvar**

The Nominal Mvar field gives the initial amount of reactive power the device would supply (in Mvars) if its terminal voltage were 1.0 per unit.

**Max Mvar, Min Mvar**

The maximum and minimum Mvar range for the switched shunt.

---

## Interface Display

---

The Interface Display is used to show the net real power (MW) flow on a group consisting of one or more of the following devices: 1) transmission lines (AC and DC) and/or transformers, 2) total tie-lines between two adjacent areas, 3) total tie-lines between two adjacent zones, 4) Generators and Loads, 5) Injection Groups, 6) multi-section lines, 7) other interfaces and 8) Contingency Actions. Interface information is useful because secure power system operation often requires that the flow on such groups be less than some limit value. For example, interface information could be used as "proxies" for other types of security constraints, such as voltage or transient stability limitations. Another major potential use for interfaces is in the [Optimal Power Flow](#) solution to monitor flows on groups of devices, such as a set of transmission lines, generation leaving a plant, load demanded in a certain region, etc. Interface information can also be extremely useful for summarizing the flows occurring on a large network. Interface flows can be monitored using the [Limit Monitoring Settings Display](#).

The Interface Display presents more detailed information for each interface that has been defined for the case. The Interface Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated interfaces. The local menu also affords the opportunity to insert new interface definitions either singly (**Insert**) or as a group ([Automatic Insertion](#)) into the model or to delete existing ones. In addition, PowerWorld has also added options for reading NERC flowgate files ([Load NERC Flowgates](#)) and writing NERC flowgate files ([Save NERC Flowgates](#)) to the local menu.

You can sort the interface information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Interface Display according to the constraints of the [area/zone/owner filters](#).

To show the interface display, select **Aggregations > Interfaces** in the [Model Explorer](#).

Interface information can be saved in the "\*.aux" auxiliary file format. See [Auxiliary Files](#) for details.

The interface Display is divided into two tabs: **Interface** and **Interface Elements**.

### Interface Elements Tab

The **Interfaces Elements tab** has a list of tabs which lists each types of element that can belong to an interface. There is a check-box below in the middle of the display which allows you to **Only show elements that belong to at least one interface**. As you select an element in the top half of the display, the bottom half of the display will list each interface which contains the selected element. Again, these provide a nice way to look through the elements that make up each interface.

### Interface Tab

The **Interfaces tab** has a list of interfaces at the top of the display. When on the Interface tab, in the bottom half of the tab is a set of subtabs which display all the elements of the presently selected interface from the top half of the display. This set of sub tabs starts with the **Elements** tab which lists all the elements of the interface regardless of the type of element. The remainder of the tabs each lists the interface elements of a specific type of interface element such as Branches, Branch Open/Close, DC Lines, Generators, Loads, Injection Groups, Multi-Section Lines and other interfaces. These subtabs provide a nice way to look through the elements that make up each interface. The top half of the Interface display contains the following fields:

#### Number



Numeric identifier for the interface.

**Name**

Alphanumeric identifier for the interface (24 characters maximum).

**Interface MW**

Current MW flow on the interface. This flow is the sum total of the Base MW Flow and the Contingent MW Flow.

**MW Limit**

Current rating for the interface in MW.

**Percent**

The actual MW flow on the interface as a percentage of the MW limit.

**Monitor Direction**

The current direction in which the MW flow is being monitored. The possibilities are From - To or To - From.

**Monitor Both**

Toggle this field to Yes in order to monitor the flow on the interface in both directions, instead of a single direction as designated by the Monitor Direction field.

**Lim MW A, Lim MW B, Lim MW C**

These three fields display the values of the three possible limits for the interface.

**Has Contingency**

This is a Yes or No field that will be set to Yes if any of the elements making up part of the interface are at a device limit.

**Contingent MW Flow**

If the interface defined contains a contingency action, the Contingent MW Flow is the approximated flow amount that would be added to the interface should the contingency occur.

**Base MW Flow**

The base MW flow is the flow on the interface prior to any considered contingency elements.

---

## Nomogram Display

---

The Nomogram Display is used to modify and create Nomogram definitions. Nomograms are combinations of two [interfaces](#) for monitoring combined flows on the interfaces concurrently. These Nomogram interfaces will have a limit definition that defines a region of allowed flow on the interfaces, and can be monitored in many tools in Simulator as potential constraints, such as the contingency analysis reporting of interface violations. Another major potential use for nomograms is in the [Optimal Power Flow](#) solution to monitor the flow on a pair of interfaces. Nomogram information can also be extremely useful for summarizing the flows on a pair of interfaces whose operation and allowed flow are closely tied together. Nomogram flows can be monitored using the [Limit Monitoring Settings Display](#).

The Nomogram Display presents more detailed information for each nomogram that has been defined for the case. The Nomogram Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information as well as view the [information dialog](#) of its associated nomograms.

You can sort the nomogram information by clicking on the heading of the field by which you want to sort. Additionally, you can choose to restrict the information shown by the Interface Display according to the constraints of the [area/zone/owner filters](#).

To show the interface display, select **Aggregations > Nomograms** in the Model Explorer.

Nomogram information can be saved in the "\*.aux" auxiliary file format. See [Auxiliary Files](#) for details.

The Nomogram display contains the following fields:

### **Name**

Alphanumeric identifier for the nomogram.

### **Int A Flow**

Current MW flow on the first interface of the interface pair forming the nomogram. This flow is the sum total of the Base MW Flow and the Contingent MW Flow.

### **Int B Flow**

Current MW flow on the second interface of the interface pair forming the nomogram. This flow is the sum total of the Base MW Flow and the Contingent MW Flow.

### **% Limit, Max Nomo-interface**

The actual MW flow on the nomogram as a percentage of the combined nomogram limit.

### **Monitor**

Indicates whether or not the nomogram should be monitored either for violations or as a constraint in the [Optimal Power Flow](#).

### **Limit Group**

The name of the limit group the nomogram is a member of for [limit monitoring](#).

---

## Injection Group Display

---

The Injection Group Records Display presents data describing each injection group in the case. This display is available by selecting **Aggregations > Injection Groups** from the [Model Explorer](#). This display is organized into two tabs with various sections available under each of the tabs. The tabs and sections provide options for displaying information about injection groups and their associated participation points in a manner in which it is most useful to the user. Injection groups can be displayed with their associated participation points or participation points can be displayed with the injection group to which they belong.

### Injection Groups Tab

This tab is organized into a top and bottom section. The top section contains the Injection Groups [Case Information Display](#) that lists all of the defined injection groups. The [local menu](#) is available by right-clicking on the display and contains options available for most case information displays as well as the option to **Show Dialog** to view the [Injection Group Dialog](#).

By default, the Injection Groups Case Information Display contains the following fields:

#### **Name**

The name of the injection group. To change the name of an injection group, simply type a new name in the corresponding cell.

#### **Number of Gens**

Identifies the number of generators contained in the injection group.

#### **% MW Gen ParFac**

Indicates the degree to which generators will contribute to the MW output of the injection group relative to loads. An injection group that has a % MW Gen ParFac value of 100% receives all of its output from generator points; an injection group that has a % MW Gen ParFac of 50% and a % MW Load ParFac of 50% receives equal MW contributions from its constituent loads and generators.

#### **Number of Loads**

Identifies the number of loads contained in the injection group.

#### **% MW Load ParFac**

Indicates the degree to which MW loads will contribute to the MW output of the injection group relative to generators. An injection group that has a % MW Load ParFac value of 100% receives all of its output from load points; an injection group that has a % MW Load ParFac of 50% and a % MW Gen ParFac of 50% receives equal MW contributions from its constituent loads and generators.

#### **% Mvar Load ParFac**

Indicates the degree to which Mvar loads will contribute to the Mvar output of the injection group relative to switched shunts. An injection group that has a % Mvar Load ParFac value of 100% receives all of its Mvar output from load points; an injection group that has a % Mvar Load ParFac of 50% and a % Mvar Shunt ParFac of 50% receives equal Mvar contributions from its constituent loads and switched shunts.

#### **Number of Shunts**

Identifies the number of switched shunts contained in the injection group.

#### **% Mvar Shunt ParFac**

Indicates the degree to which switched shunts will contribute to the Mvar output of the injection group relative to Mvar loads. An injection group that has a % Mvar Shunt ParFac value of 100% receives all of its Mvar output from switched shunt points; an injection group that has a % Mvar Shunt ParFac of 50% and a % Mvar Load ParFac of 50% receives equal Mvar contributions from its constituent loads and switched shunts.

#### **Total MW Injection**

Indicates the current total MW injection of all elements in the injection group.

#### **Total Mvar Injection**

Indicates the current total Mvar injection of all elements in the injection group.

The bottom section contains additional tabs: **Participation Points (All)**, **Generators**, **Loads**, **Switched Shunts**, and **Injection Groups**. These tabs contain tables, which are variations of the [Participation Point Records Display](#), that list all of the individual participation points for the selected injection group by element type. Participation points can be added, deleted, or modified for the selected injection group from any of the tables on these tabs. The **Participation Points (All)** table lists only the fields that are relevant for defining participation points while the specific element tables also make available the fields that are specific to that element type.

The **Generators**, **Loads**, **Switched Shunts**, and **Injection Groups** tabs have the option to **Show points that are contained by other injection group points**. This option is only relevant if an injection group contains a participation point that is another injection group. When this option is checked, the individual participation points that are contained in the injection group point will also be displayed. The **Contained by** field will list the name of the injection group that contains a particular participation point. When this option is not checked, the points displayed will be points that are contained in the selected injection group only and no additional information will be provided about the points contained inside any injection group points belonging to the selected injection group. If the **Contained by** field is empty, this means that the participation point is explicitly defined with the selected injection group and does not belong to it through an injection group point.

### **Participation Points Tab**

This tab is organized into a top and bottom section. The top section contains tabs for **Generators**, **Loads**, **Switched Shunts**, and **Injection Groups**. Tables on these tabs will list power system elements of each particular type. If the option to **Only show elements that belong to at least one injection group** is not checked, then all elements of each type in the case will be listed on one of these tabs. If this option is checked, then only those elements that belong to an injection group will be listed. The tables on these tabs are [Case Information Displays](#) for each particular type and have the full functionality of case information displays.

The bottom section is an Injection Groups Case Information Display. When an element from one of the tables in the top section of the tab is selected, this table will be populated with the list of injection groups to which the selected element belongs.

---

## Injection Group Dialog

---

The Injection Group Dialog provides information about injection groups and allows their modification. Specifically, the Injection Group Dialog lists the number of generators, loads, and switched shunts contained in the group and the percentage contribution of generators, loads, and switched shunts to the injection group's output. The Injection Group Dialog also houses the Participation Points Records display, from which points can be added and deleted from the injection group's list of participants and the various attributes of the points can be changed.

To view the Injection Group Dialog for a particular injection group, open the [Injection Group Display](#) by selecting **Aggregations > Injection Groups** from the [Model Explorer](#). Find the injection group of interest in the Injection Groups Case Information Display and right-click on it. Then select **Show Dialog** from the resulting local menu.

The Injection Group Dialog contains the following fields and controls:

### Name

Identifies the injection group whose information is currently displayed. Selecting a different name from this dropdown box will display information for another injection group.

### New, Delete

To insert a new injection group from this dialog, press **New**, and supply the name for the new injection group. To delete the injection group that is currently being shown, click **Delete**.

### Save

Changes made to injection groups through the Injection Group Dialog are not immediately saved with the power system case. This allows the option of canceling any changes without impacting the case. Click the **Save** button to store any updates with the case.

### # Gens

Displays the number of generators contained in the injection group.

### # Loads

Displays the number of loads contained in the injection group.

### # Shunts

Displays the number of switched shunts contained in the injection group.

### % MW Gen Part., % MW Load Part., % MVR Load Part., % MVR Shunt Part.

Displays the relative contributions of generators, loads, and switched shunts to the output of the injection group.

### OK, Cancel

Changes made to the injection groups through the Injection Group Dialog are not immediately saved with the power system case. This allows the option of canceling any changes without impacting the case. Click **Cancel** to close the dialog without saving the changes. Click **OK** to close the dialog and save the changes.

## Participation Points

The tab on the bottom of the Injection Group Dialog lists the points that make up the injection group. This display is called the [Participation Point Records Display](#). By right-clicking on this display, points can be added and deleted from the injection group. Properties of specific points

can be changed by entering changes directly in the table. The **Insert Points** button will open the [Add Participation Points Dialog](#) that can also be used for modifying the points.

---

## Participation Point Records Display

---

The Participation Point Records Display shows information about the points that comprise a particular injection group. This display can be accessed for a particular injection group from the [Injection Group Dialog](#) or the [Injection Group Display](#).

The Participation Point Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which participation points can be deleted or added to the injection group through the [Add Participation Points Dialog](#). Selecting **Insert** from the local menu will open the Add Participation Points Dialog that will allow the addition, deletion, and modification of participation points in the injection group.

The Participation Point Records Display shows the following fields by default:

### Point Type

Every participation point is a generator (GEN), load (LOAD), switched shunt (SHUNT), or injection group (INJECTIONGROUP).

### Number

Identifies the number of the bus to which a generator, load, or switched shunt is connected.

### Name

Identifies the name of the bus to which a generator, load, or switched shunt is connected.

### ID

Identifies the ID of the generator, load, or switched shunt or the name of the injection group.

### AutoCalc?

If the value of **AutoCalc** is *YES*, the participation factor of the point is re-calculated with every use to be consistent with the way the point's participation factor was initially defined. If the value of **AutoCalc** is *NO*, the participation factor is assumed fixed at its present value.

### Initial Value

Indicates how the participation factor of the point was originally computed. For generators, the possible values of this field are

**SPECIFIED** The participation factor was specified as a constant.

**MAX GEN INC** The participation factor was defined as the difference between the generator's maximum MW output and its present MW output.

**MAX GEN DEC** The participation factor was defined as the difference between the generator's present MW output and its minimum MW output.

**MAX GEN MW** The participation factor was defined as the maximum MW output of the generator.

For loads, the Initial Value property can assume only two possible values:

**SPECIFIED** The participation factor was specified as a constant.

**LOAD MW** The participation factor was defined as the size of the load in MW.

For switched shunts, the Initial Value property can assume four possible values:

**SPECIFIED** The participation factor was specified as a constant.

**MAX SHUNT INC** The participation factor was defined as the difference between the maximum Mvar output of the switched shunt and its present nominal Mvar output.

**MAX SHUNT DEC** The participation factor was defined as the difference between the present nominal Mvar output and the minimum Mvar output of the switched shunt.

**MAX SHUNT MVAR** The participation factor was defined as the maximum Mvar output of the switched shunt.

For injection groups, the Initial Value property can assume only one possible value:

**SPECIFIED** The participation factor was specified as a constant.

The **Initial Value** field is important if saving the injection groups to an auxiliary file and using them with another case that might have a different generation dispatch or load profile. If the **Initial Value** for a point is specified as *GEN MAX INC*, for example, and the point is loaded from an auxiliary file into another case, Simulator will re-calculate the point's participation factor to match the generator's positive MW reserve in that case.

The **Initial Value** field is also important if the **AutoCalc** field is set to *YES*, because **AutoCalc** uses the rule defined by the **Initial Value** field to recalculate the participation factor with every use. For example, if the **Initial Value** is *MAX GEN INC* and **AutoCalc** is *YES*, the point's participation factor will be updated to match the generator's MW reserve every time the point is accessed.

When the value of the **Initial Value** field is toggled, the point's participation factor, shown in the **ParFac** field, will update to match the new definition. The **Initial Value** field must equal *SPECIFIED* to be allowed to change the value of the point's **ParFac** field by typing the new value directly into the field.

If **AutoCalc** is *NO* and the injection group is not intended to be used with any other case, then the **Initial Value** field should either be ignored or set to *SPECIFIED*.

### ParFac

Indicates the participation factor of the participation point. The participation factor defines the relative contribution of the point to the total output of the injection group. The load, generation, or switched shunt change associated with each point is calculated based on the value of the participation factor, with values for points having the largest participation factors experiencing the greatest change.

The participation factor for injection group participation points indicates the relative contribution of the entire injection group to the total output of the injection group to which it belongs. The participation factors of the individual points in the injection group point further define the relative contribution of these points to the total output of the injection group. The following two injection groups provide an example of how injection group points impact the injection group to which they belong when the points are normalized for use:

#### Injection Group 1    Injection Group 2

Gen 1, ParFac = 50    Gen 5, ParFac = 40

Gen 2, ParFac = 30    Load 1, ParFac = 20

InjectionGroup 2, ParFac = 20    Load 2, ParFac = 20

Total = 100    Total = 80

When Injection Group 1 is used, the participation factors are normalized so that each element provides the following contribution:



Gen 1, Normalized ParFac =  $50/100 = 0.5$

Gen 2, Normalized ParFac =  $30/100 = 0.3$

Gen 5, Normalized ParFac =  $20/100 * 40/80 = 0.1$

Load 1, Normalized ParFac =  $20/100 * 20/80 = 0.05$

Load 2, Normalized ParFac =  $20/100 * 20/80 = 0.05$

Total Normalized = 1.0

To add points to the injection group, right-click on the Participation Point Records Display and select **Insert** from the local menu. This opens the [Add Participation Points Dialog](#).

---

## Island Display

---

The Island Display presents information on the system's islands. An island consists of a group of buses that are electrically connected via ac transmission lines and transformers and thus operate in synchronism with one another. Multiple islands can be connected together via dc transmission lines. Each island requires a slack bus. All systems have at least one island, which may encompass the entire system.

The Island Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, and modify its information. You can also sort the islands information by clicking on the heading of the field by which you want to sort.

To show this display select **Aggregations > Islands** in the [Model Explorer](#). The display is only available in run mode.

You cannot modify any fields on this display.

The display has the following fields for each island:

**Slack Bus Number, Slack Bus Name, Slack Bus Area**

Number, name and area of the slack bus for the island. Each island requires at least one slack bus.

**Total Buses**

Total number of buses in the island.

**Energized**

Indicates whether the island is connected to a source of power.

**Gen MW, Gen Mvar**

Total real and reactive generation for the island.

**Load MW, Load Mvar**

Total real and reactive load for the island.

**Scheduled Exports**

The power scheduled to be provided by the island to other regions of the system. Because the island is isolated from the rest of the system, this export requirement is currently not met.

**DC MW Exports**

The power provided by the island to other regions of the system across a DC line.

---

## MW Transactions Display

---

The MW Transactions Display is a quick way to view the defined base transactions between areas within the load flow case.

The **Matrix of Transactions** display is set up as a matrix of transactions, with the areas listed as both the column and row identifiers. It is important to note that the direction of the transaction is such that the area represented by the **row** is assumed the exporting area, and the area represented by the **column** is assumed the importing area. For example, if there is a 50 MW transaction from area one to area two, you will see a +50 in the matrix in row 1, column 2. However, if you look at row 2, column 1, you will see a – 50. This is because the grid is displaying the EXPORT from area two to area one, but since area two is importing, not exporting, the value is represented as negative.

Each row and column position, with the exception of the diagonal positions (it does not make sense for an area to export to itself), can be directly modified by the user in this information display. As you type a value in one of the matrix positions, Simulator automatically fills the symmetric matrix position with the negative of the value you enter. This makes for a quick and easy location for adding and removing transactions from the case.

Note that you cannot modify **Unspecified** transactions in the MW Transactions display. This makes sense, as the unspecified transactions have only one associated area. To modify unspecified transaction amounts, you must open the [Area Information Dialog](#) for the area you wish to modify, and change the Unspecified transaction amount.

The **List of Transactions** is just another way of showing the transactions between areas. The List of Transactions Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which you can print, copy, or modify its records as well as view the [information dialog](#) of its associated areas. You can also sort the area records by clicking on the heading of the field by which you want to sort.

Note that a special feature is available in the right-click local menu called **Clear Transactions and auto-insert tieline transactions**. By choosing this option all MW transactions in the case will be deleted, and all Unspecified MW transactions for each area will be set to zero. Then new MW transactions will be created between each pair of areas that area directly connect to one another. The amount of the new MW transactions will be set equal to the actual sum of the flow on the tielines between the connected areas.

By default, the List of Transactions display contains the following fields:

### **Export Area Number, Export Area Name**

Exporting area number and alphanumeric identifier.

### **Other Area Number, Other Area Name**

Importing area number and alphanumeric identifier.

### **MW Transfer**

Value of the transfer in MW.

---

## Transaction Dialog

---

The Transaction Dialog can be used to modify or create Base Interchange Transactions between two areas. This dialog can be opened by right-clicking in the Base Interchange table of the [Area Information Dialog](#) and choosing Show Dialog to see an existing transaction definition, or Insert to add a new transaction.

The transaction dialog is divided into two pages of controls:

### Information

#### Transacting Area

This is the "from" area for the transaction. For an export from this area, the transaction value will be positive. For an import into the Exporting Area, the transaction value specified would be negative. Flow out (export) of the exporting area is always considered positive.

#### Transaction To Area

This is the "to" area for the transaction.

#### Transaction ID

New in Simulator version 10 is the ability to have multiple transactions defined between the same two areas. Because of this, it is now required that transactions also have a Transaction ID.

#### Rename Transaction ID

If you wish to change the transaction ID for a particular transaction, enter the new value in the **Transaction ID** field, and press this button.

#### Switch Directions

Press this button if you wish to reverse the defined Exporter and Importer for the transaction.

#### Transaction MW Amount

The MW amount of the transaction being defined. This value should be positive for an export from the Exporting area to the Importing area. The value can also be entered as negative to define a transaction into the Exporting area from the Importing area.

#### Transaction Minimum MW

The minimum transaction amount between the two areas. This field is only enabled if the check box labeled **Transaction Dispatchable in OPF** is checked.

#### Transaction Maximum MW

The maximum transaction amount between the two areas. This field is only enabled if the check box labeled **Transaction Dispatchable in OPF** is checked.

#### Exports/Imports Transmission Charge

The cost to transfer power, in \$/MWh. This adds an economic penalty for making the transfer, making the transfer less likely to take place. Half the charge is assigned to the buyer and half to the seller.

#### Transaction Enabled

Transaction can now be defined and either enabled or disabled. Any disabled transactions will be ignored in both a standard power flow solution and an OPF solution.

#### Transaction Dispatchable in OPF

Checking this box enables the transfer to be dispatched by the OPF algorithm. Dispatching the transaction makes the two areas of the transaction appear to be one area for the purpose of economically dispatching the generation in the two areas. The transaction can have a maximum and minimum transfer amount when dispatchable, and a transmission charge associated with the transaction.

#### **Determine Price in OPF**

Checking this box allows the OPF algorithm to determine the cost associated with the transfer. The cost is determined by the marginal cost of enforcing the power balance constraint for the combined areas. This is the typical way to implement a transfer if *both* areas are on OPF control. If only one of the two areas are on OPF control, then the area which is off of OPF control needs to specify a price for the transfer. This is done by explicitly defining a piecewise linear cost curve.

#### **Piecewise Linear Transaction Cost Curve**

These two curves are only enabled if the option **Transaction is Dispatchable in OPF** is checked and the option **Determine Price in OPF** is unchecked. These two curves can be defined for the purpose of assigning a price to the transfer of power between one area on OPF control and another area which is not on OPF control. Separate curves can be defined for export transactions (from the Exporter to the Importer) and import transactions. To add points to the curves, simply right-click in the grid and choose Insert from the local menu. Enter the MW value and corresponding marginal cost for the inserted breakpoint of the piecewise linear curve you are defining. To delete a point, right-click on that row in the grid and choose Delete from the local menu.

#### **Custom**

The Custom page of the Transaction dialog is simply a location to log information about the transaction in the Memo box, and to see the values (if any) stored in some of the custom fields for the transaction. To log information about the transaction, simply switch to the Custom page on the dialog, and start typing your information or comments about the transaction in the page, or enter custom values in one of the custom fields.

---

## Limit Monitoring Settings

---

Simulator offers many tools to study the capabilities of a power system. Examples include:

- [Contingency Analysis](#)
- [Available Transfer Capability \(ATC\)](#)
- [Optimal Power Flow \(OPF\)](#)
- [Security Constrained OPF \(SCOPF\) Overview](#)
- [PV Curve and QV Curve Tool](#)

All of these tools make extensive use of power system limits. Limits for various power system elements include:

- MVA (or Amp) limits on transmission lines and transformers
- MW limits on Interfaces
- High and low voltage limits for Buses

The accuracy of all these limits is very important, as is the specification of which limits should be monitored. While ensuring the accuracy of input data such as power system limits must be left to the user, PowerWorld Simulator provides several ways to specify which limits should be monitored. Limit Monitoring is specified according to settings for the Area, Zone and Limit Group to which the power system element belongs. A power system element is monitored only if ALL of the following conditions are met.

### Conditions for Monitoring an Element's Limit

- Its Monitor field is set to **YES**
- Its Limit Group is Enabled
- Its Area is set to Report Limits and it meets the KV range for reporting
- Its Zone is set to Report Limits and it meets the KV range for reporting

The [Limit Monitoring Settings Dialog](#) gives you ability to specify all these settings to setup the limits you want to monitor, and thus enforce in the various tools that Simulator provides.

---

## Limit Monitoring Settings and Limit Violations Dialog

---

The Limit Monitoring Settings Dialog gives you ability to specify which limits you want to monitor, and thus enforce in the various tools that Simulator provides. In general, keep in mind that a bus, transmission line/transformer, or interface's limit is monitored only if the following conditions are met.

### Conditions for Monitoring an Element's Limit

- Its Monitor field is set to **YES**
- Its Limit Group is Enabled
- Its Area is set to Report Limits and it meets the KV range for reporting
- Its Zone is set to Report Limits and it meets the KV range for reporting

The following fields are shown on the display:

### Elements To Show

Change this value to modify which elements are displayed in the Buses, Lines, and Interfaces. Set it to...

**All Elements** to show all Buses, Lines and Interfaces regardless of monitoring settings.

**Monitored Elements** to only display Buses, Lines and Interfaces that meet the conditions for monitoring. See [Limit Monitoring Settings](#) for more information on setting which values are monitored.

**Violating Elements** to only display Buses, Lines and Interfaces that are violating. See [Limit Monitoring Settings](#) for more information on setting what is considered a violation.

### Number of Violations

#### High Voltage Buses and Low Voltage Buses

Shows the total number of bus voltage magnitude violations. These violations are shown on the **Bus Voltage Magnitude Limit Violations Display**. This field is read-only.

#### Low Voltage Suspects

Shows the total number of buses whose voltages have fallen below a designated threshold to indicate a low voltage solution is being reached in some location in the system.

#### Line/Transformer Violations

Shows the total number of violations reported on **Line and Transformer Limit Display**. This field is read-only.

#### Interface Violations

Indicates the total number of interface violations. These violations are shown on the **Interface Violations Display**. This field is read-only.

### Limit Group Values

This part of the dialog shows information about the Limit Groups with greater description shown below.

### Save/Load Monitoring Settings

The Save and Load Monitoring Settings buttons allow the user to save the current set or load a previously saved set of Limit Monitoring Settings options. The options are saved in a [PowerWorld Auxiliary File](#) text format.

### Do not monitor radial lines and buses

Check this box to ignore limits on radial lines and buses throughout the case. In this instance a radial bus is defined as a bus that is connected to the rest of the power system by a single inservice transmission line. A radial line is the defined as a line connected to a radial bus. This means that in a 5 lines in series that connect to a radial load, only the last bus and last line will be considered radial.

If you would like to better determine which lines are "radial" in a more traditional sense, use the feature [Branches that Create Islands](#) that allows you to easily generate a list of branches that if taken out of service will split an existing island into two. These represent lines which may be considered radial.

### Tabs on the Limit Monitoring Dialog

The tabs on the Limit Monitoring Settings Dialog allow you to change these settings. These tabs contain tables showing lists of the respective elements. Since these tables are another variety of the [Case Information Displays](#), you may interact with it in a familiar manner. Click on any of the field headings to sort by that field. Right-click on the display to call up the display's [local menu](#). From the local menu, you can print the violations, [copy the violation records](#) to the Windows clipboard for use with another application, modify the format and content of the violations listing, view the information dialog of the respective element, and view the [Quick Power Flow List](#) or [Bus View Display](#).

### Buses Tab

Each tab shows a list of the respective type of power system element. The important columns include

**Number, Name** Bus number between 1 and 99,999, and its alphanumeric identifier.

**Area Name** Alphanumeric identifier of the bus' area.

**Monitor** : toggle this between **YES** and **NO** to set whether the specific element should be monitored.

**Limit Group** : toggle this value to specify which Limit Groups the element belongs to

**PU Volt** Bus' per unit voltage magnitude.

**Volt (kV)** Bus' actual voltage magnitude in kV. This is the per unit voltage magnitude multiplied by the bus' nominal voltage.

Note: If Elements to Show is set to Monitored Elements, then only elements which meet the conditions for monitoring will be displayed in these lists.

### Lines Tab

Each tab shows a list of the respective type of power system element. The important columns include

**From Bus Number and Name** "From" bus number and name. For transformers, the from bus is the tapped side. Right- clicking on either of these fields allows you to see all the flows measured at the "from" bus using the [Quick Power Flow List](#) or [Bus View Display](#) local menu options.

**To Bus Number and Name** "To" bus number and name. For transformers, the to bus is the untapped side. Right-clicking on either of these fields allows you to see all the flows into the "to" bus using the [Quick Power Flow List](#) or [Bus View Display](#) local menu options.



**Circuit** Two-character identifier used to distinguish between multiple lines joining the same two buses.

**Monitor** : toggle this between **YES** and **NO** to set whether the specific element should be monitored.

**Limit Group** : toggle this value to specify which Limit Groups the element belongs to

**Used Limiting Flow, Limit, Used % of Limit** The flow at the end of the branch selected for measurement, and its MVA or Amp limit. The percentage equivalent of the flow to it's limit is given in the Used % of Limit column.

**MVA or Amps?** Units used with the Max Flow and the Limit field. All flows are expressed either in MVA or amps.

Note: If Elements to Show is set to Monitored Elements, then only elements which meet the conditions for monitoring will be displayed in these lists.

### Interfaces and Nomograms Tab

Similar to the Buses and Lines Tabs shown above. The important columns are the **Monitor** and **Limit Group** columns

### Area Reporting and Zone Reporting Tabs

These tabs display all the Areas and Zones in the system. The important columns are

**Report Limits:** toggle this between **YES** and **NO** to set whether the specific element should be monitored.

**Report Min kV and Report Max kV:** Only buses and lines within this kV range will be monitored.

### Modify/Create Limit Group Tab

Every power system elements belongs to a single Limit Group. By default, all power system elements are in the same Limit Group that is named "**Default**". New limit groups can be added by right-clicking in the table and choosing **Insert** from the popup menu. Additional limit groups give the flexibility of assigning devices to different groups, where each group can have its own set of defined limit information.

The limit group stores important values regarding the enforcement and monitoring of the power system elements within the Limit Group. The values are

**Branch Percentage:** The percentage to which Simulator's study tools will limit a line or transformer. Typically this is 100%, but it can be modified. In [Contingency Analysis](#), then lines will be flagged as violated if they exceed this percentage. In performing an [Optimal Power Flow](#), all attempts will be made to keep the line below this percentage.

**Line Rate Set:** You may define eight different ratings to transmission lines or transformers (for more information see [Line/Transformer Information](#)). Change this value to specify which rating set should be used for lines/transformers in the limit group.

**Amps or MVA:** Limits for transmission lines and transformers are always entered in MVA. However, when reporting limit violations, it is common to check transmission line limits in terms of their amp loading. If the Treat Line Limits As Equivalent Amps is checked, the limits for transmission lines are reported in amps rather than MVA. If this box is not checked, limits for both transmission lines and transformers are expressed in MVA.

For reference, note that the amp rating of a line is derived from the MVA rating using the formula

$$AmpRating = \frac{MVARating}{\sqrt{3} * BaseKV} * 1000$$

**Interface Percentage:** The percentage to which Simulator's study tools will limit an interface. Typically this is 100%, but it can be modified. In [Contingency Analysis](#), then interfaces will be flagged as violated if they exceed this percentage. In performing an [Optimal Power Flow](#), all attempts will be made to keep the interface below this percentage.

**Interface Rate Set:** You may define eight different ratings to an interface. Change this value to specify which rating set should be used for interfaces in the limit group.

**Nomogram Percentage:** The percentage to which Simulator's study tools will limit a nomogram.

**Low PU Volt:** Buses will be flagged as violated if they fall below this per unit voltage.

**High PU Volt:** Buses will be flagged as violated if they go above this per unit voltage.

**Disabled:** Set the value to **YES** to ignore all power system element limits in the Limit Group. Set the value to **NO** to monitor limits according to the settings of the Limit Group.

**Contingency Line Rate Set:** This field specifies the rating set used for post-contingency monitoring of Lines/Transformers.

**Interface Contingency Rate Set:** This field specifies the rating set used for post-contingency monitoring of Interfaces during Contingency Analysis.

**Limiting End:** Specified as higher or lower, this field determines whether the higher or the lower flow amount on the element is used for reporting a limit violation. If you use higher, it is possible that the lower flow is not violating the line limit. If you use lower, then you are guaranteed that the element limit is being violated at both ends of the element.

**Use Limit Cost:** If this field is set to yes, you are enabling the capability to have a cost function associated with enforcing constraints. This cost function can be viewed as similar to a generator cost function, in that as the constraint becomes overloaded by larger amounts, the marginal cost of enforcing the constraint will increase. For setting the piecewise linear limit cost curve, right-click and choose show dialog from the popup menu to access the Limit Group Dialog.

Often monitoring elements according to Area and/or Zone is all that you need. However, if you need to monitor specific groups of power system elements and not others then you need to create some new Limit Groups.

---

## Limit Group Dialog

---

The Limit Group Dialog reflects much of the same information that is summarized in the [Limit Monitoring Settings](#) Dialog's Modify/Create Limit Groups page. The same information can be changed here as on the previously mentioned display. One important feature that is unique to this dialog is the ability to define a piecewise linear limit cost curve, which can be used for setting up "soft" constraints used in the [Security Constrained Optimal Power Flow](#).

### Disabled

Check this option to ignore all power system element limits in the Limit Group. Leave the box unchecked to monitor limits according to the settings of the Limit Group.

### Add New Limit Group

Clicking this button will create a new limit group, displaying a dialog in which the name for the new limit group can be specified.

### Rename Limit Group

Clicking this button will display a dialog in which a new name for the current limit group can be specified.

## Lines/Interfaces

### % of Limit for Reporting (Branches)

The percentage to which Simulator's study tools will limit a line or transformer. Typically this is 100%, but it can be modified. In [Contingency Analysis](#), lines will be flagged as violated if they exceed this percentage. In performing an [Optimal Power Flow](#), all attempts will be made to keep the line below this percentage.

### Line/Transformer Rating Set

You may define eight different ratings to transmission lines or transformers (for more information see [Line/Transformer Information](#)). Change this value to specify which rating set should be used for lines/transformers in the limit group.

### Line/Transformer Contingency Rating Set

This field specifies the rating set used for post-contingency monitoring of Lines/Transformers.

### Treat Transmission Line Limits as Equivalent Amps

Limits for transmission lines and transformers are always entered in MVA. However, when reporting limit violations, it is common to check transmission line limits in terms of their amp loading. If the Treat Line Limits As Equivalent Amps is checked, the limits for transmission lines are reported in amps rather than MVA. If this box is not checked, limits for both transmission lines and transformers are expressed in MVA.

For reference, note that the amp rating of a line is derived from the MVA rating using the formula

$$AmpRating = \frac{MVARating}{\sqrt{3} * BaseKV} * 1000$$

### Limiting End of Line

Specified as higher or lower, this field determines whether the higher or the lower flow amount on the element is used for reporting a limit violation. If you use higher, it is possible

that the lower flow is not violating the line limit. If you use lower, then you are guaranteed that the element limit is being violated at both ends of the element.

#### **% of Limit for Reporting (Interfaces)**

The percentage to which Simulator's study tools will limit an interface. Typically this is 100%, but it can be modified. In [Contingency Analysis](#), interfaces will be flagged as violated if they exceed this percentage. In performing an [Optimal Power Flow](#), all attempts will be made to keep the interface below this percentage.

#### **Interface Rating Set**

You may define eight different ratings to an interface. Change this value to specify which rating set should be used for interfaces in the limit group.

#### **Interface Contingency Rating Set**

This field specifies the rating set used for post-contingency monitoring of Interfaces during Contingency Analysis.

#### **% of Limit for Reporting (Nomograms)**

The percentage to which Simulator's study tools will limit a nomogram.

#### **Use Limit Cost**

If this field is checked, you are enabling the capability to have a cost function associated with enforcing constraints. This cost function can be viewed as similar to a generator cost function, in that as the constraint becomes overloaded by larger amounts, the marginal cost of enforcing the constraint will increase.

Once the box is checked, the table for defining the limit cost function will become enabled. The starting point (% Flow) must be at or above 100%. You can then begin inserting additional points in the piecewise linear curve by right-clicking in the table and selecting Insert Point from the popup menu, followed by entering the new percent flow and marginal cost. Note that the cost function must be strictly increasing, meaning the next marginal cost value must be equal to or greater than the immediately previous value.

When constraints are given the ability to use this limit cost curve, it effectively gives the optimal power flow the ability to "dispatch" the limit of the elements according to the marginal costs of the limit cost curve. You are determining the point on the limit cost curve where the shadow price of enforcing the constraint is met. Thus the OPF routine will determine how much you will allow the element to be overloaded by giving the element the piecewise linear curve. The OPF will solve the problem, and optimize the constraints as it determines shadow prices for each constraint and adjusts the limit accordingly during the iterations of the routine. Since these are somewhat flexible limit assignments to the elements, they are sometimes considered "soft" constraints.

### **Buses/Summary**

#### **Low Per Unit Limit**

Buses will be flagged as violated if they fall below this per unit voltage.

#### **High Per Unit Limit**

Buses will be flagged as violated if they fall below this per unit voltage.

#### **Low-voltage solution flag**

Enter a per unit value at which you wish to measure a voltage and consider the solution to be a low voltage solution.

**Include Out-Of-Service Buses**

Check this check-box to include out-of-service buses.

**Elements using Limit Group**

Lists the number of elements that are members of the selected limit group.

---

### **Owner Data Information Display**

---

The Owner Data display will display the owners sorted by owner number, and show the total number of devices, as well as a breakdown of the number of specific devices. The owner display will also summarize the total device output or demand for the owned devices, such as generator output and load demand.

You can right click on the owner grid to open the local menu, from which you can perform various actions and view an informational regarding the owner selected in the grid. Once an [owner dialog](#) is displayed for a particular owner, you can browse through information on all the owners.

---

## Owner Dialog

---

The Owner Dialog displays summary information about the devices designated as owned by the selected owner.

The dialog for an individual owner displays each of the elements for that owner, along with the percentage of ownership, in the device pages labeled [Buses](#), [Loads](#), [Generators](#), and [Lines](#).

The dialog also contains a [memo](#) page for making comments or notes about the selected owner.

### General Info

#### Load and Generation

The information in the Load and Generation section provides a summary of the total injections of the owned devices. This includes a total of all the load, generation, and shunt injections owned.

#### Summary of Owner Objects

This information section simply lists a total number of all owned devices. Note that buses and loads do not currently have fractional ownership. However, generators and transmission lines do have the capability to be partially owned by more than one owner. Therefore it is possible to see fractional ownership amounts for the number of generators or lines.

#### Generator Costs and OPF Results

This information provides a summary of LMP and cost information determined by running an [Optimal Power Flow](#) solution. The information provided is determined only for devices owned by the selected owner.

---

### **Owned Bus Records Display**

---

The Owned Bus Records display is a [Case Information Display](#) that allows you many of the same options as the Case Information [Bus Display](#). In addition to the information available for viewing from the [Bus Display](#), the Owned Bus Records Display also shows the owner number and percentage of ownership. Currently, Simulator only allows one owner per bus, so by default the ownership percent is always 100%.



---

**Owned Load Records Display**

---

The Owned Load Records display is a [Case Information Display](#) that allows you many of the same options as the Case Information [Load Display](#). In addition to the information available for viewing from the [Load Display](#), the Owned Load Records Display also shows the owner number and percentage of ownership. Currently, Simulator only allows one owner per load, so by default the ownership percentage is always 100%.

---

### **Owned Generator Records Display**

---

The Owned Generator Records Display is a [Case Information Display](#) that allows you many of the same options as the Case Information [Generator Display](#). In addition to the information available for viewing from the [Generator Display](#), the Owned Generator Records Display also shows the owner number and percentage of ownership. Currently, Simulator allows up to four owners for one generator, with a total percentage ownership between 0 and 100%. To make changes to the owners and ownership percentages for a generator, right-click on the grid and choose Show Dialog and select the [Owners, Area, Zone tab](#).

---

**Owned Line Records Display**

---

The Owned Line Records Display is a [Case Information Display](#) that allows you many of the same options as the Case Information [Line and Transformer Display](#). In addition to the information available for viewing from the [Line and Transformer Display](#), the Owned Line Records Display also shows the owner number and percentage of ownership. Currently, Simulator allows up to four owners for one generator, with a total percentage ownership between 0 and 100%. To make changes to the owners and ownership percentages for a transmission line, right-click on the grid and choose Show Dialog and select the [Owners tab](#).

---

## **Jacobian Display**

---

The Jacobian display is a matrix showing the system Jacobian matrix for the currently loaded Simulator case. This display can be very useful for educational purposes. Keep in mind that for a large case, this display can contain a very large matrix. It is possible to right-click on this display and save the grid to a Matlab formatted file, or to export the grid to an Excel spreadsheet. Be aware that a Jacobian matrix from a large will often exceed the size limitations of an Excel spreadsheet.

---

**Ybus Display**

---

The Ybus display (bus admittance matrix) is a matrix showing the system Ybus for the currently loaded Simulator case. This display can be very useful for educational purposes. Keep in mind that for a large case, this display can contain a very large matrix. It is possible to right-click on this display and save the grid to a Matlab formatted file, or to export the grid to an Excel spreadsheet. However, Excel does have limitations on the number of rows and columns that could quickly be exceeded with a Ybus from a large case.

---

## **Chapter 9: Injection Groups**

---

This chapter describes the setup and use of injection groups. Injection groups are useful for modeling generators and loads together like a unit.

The following topics are covered in this chapter:

- Injection Group Overview
- Working with Injection Groups
- Participation Groups

---

## Injection Group Overview

---

An injection group is a collection of loads, generators, switched shunts, and/or other injection groups. In that respect, injection groups are somewhat analogous to areas and zones. However, unlike with areas and zones, generators, loads, switched shunts, and injection groups can belong to more than one injection group. Moreover, a single injection group may contain generators, loads, and switched shunts from several different areas and zones. Thus, injection groups are useful when modeling a collection of generators, loads, and switched shunts that act together as a unit, regardless of each individual's area or zone affiliation. The most common use for injection groups is to model a transfer of power from one group of generators and loads to another for [PTDF](#) calculations and for [PV/QV analysis](#). They are called **injection groups** because their components (generators, loads, and switched shunts) are objects that inject power into the network.

---

## Creating Injection Groups

---

Injection groups can be created from the [Injection Group Display](#), [Injection Group Dialog](#), [Generator Display](#), [Load Display](#), [Switched Shunt Display](#), or by selecting a group of display objects on the oneline display.

To create an injection group from the Injection Group Display, select **Aggregation > Injection Groups** in the [Model Explorer](#) to open the [Injection Group Display](#). This display is organized into two tab pages with additional tabs and displays available under each. The Injection Groups tab lists and provides a summary of all defined injection groups. Click the right mouse button on this display, and select **Insert** from the resulting [local menu](#) and the [Injection Group Dialog](#) will open with a default name assigned to the new injection group. The Injection Group Dialog allows the addition of participation points to the new injection group and allows updating existing injection groups. The **Load** option on the local menu of the Injection Group Display can also be used to import injection groups from an Injection Group Auxiliary Data File and the **Auto Insert Injection Groups** option can be used to automatically insert injection groups based on options set in the [Auto Insertion of Injection Groups Dialog](#).

If the [Injection Group Dialog](#) is already open, clicking the button labeled **New** will create a new injection group. A prompt will then ask for the name of the injection group to add.

To create an injection group from the [Generator Display](#), [Load Display](#), or [Switched Shunt Display](#), select the elements to add to the injection group from one of these displays, right-click on the selection to bring up the local menu, and select **Create Injection Group from Selection** from the local menu. The [Injection Group Dialog](#) will open with the selected elements added to a new injection group. Any necessary modifications to the group can be made from the Injection Group Dialog.

To create an injection from a selection of display objects, select the objects to add to the injection group. Only generators, loads, and switched shunts will be added to an injection group, but if other objects are selected, the injection group will still be created. The objects that cannot be added to an injection group will simply be ignored. Once all objects are selected, right-click on the selection to bring up the local menu. On the local menu select **Create Injection Group from Selection**. The [Injection Group Dialog](#) will open with the selected objects added to a new injection group. Any necessary modifications to the group can be made from the Injection Group Dialog. More than one object must be selected on the display for the create injection group option to be available.



---

## Auto Insert Injection Groups

---

Many times injection groups need to be created comprised of generators, loads, or switched shunts grouped by areas, zones, super areas, owners, or some other criteria. Injection groups can be created automatically by setting the grouping options by using the Auto Insertion of Injection Groups Dialog.

The Auto Insertion of Injection Groups Dialog can be accessed by selecting **Records > Auto Insert Injection Groups...** from the case information toolbar on an [Injection Group Display](#). The following options are available:

### Element Type

This option is used to select the type of element, **Generators, Loads, or Switched Shunts**, to add to the new injection groups. Only a single type of element can be added to the new injection groups.

### Group By

This option dictates the common property by which all elements in the injection group will be grouped. Elements can be grouped by **Areas, Zones, Super Areas, Owners**, or a selected **Custom Field**. If for example the Area option is selected, then injection groups will be created for each of the defined areas in the case. The same holds true for Zones, Super Areas, and Owners. If the **Custom Field** option is selected, then injection groups will be created for all elements whose Custom Field value is the same. For example, if the Custom Field option is selected and Cust Float 1 is set as the field to use, then all of the elements whose Cust Float 1 field are the same will be grouped together. If there are elements whose Cust Float 1 field is set to 1 and there are elements whose Cust Float 1 field is set to 2, then two new injection groups will be created. One will contain all of the elements whose Cust Float 1 field is 1 and the other will contain all of the elements whose Cust Float 1 field is 2. Elements whose selected Custom Field is blank will not be included in any of the new injection groups.

For **Areas, Zones, Super Areas**, and **Owners**, the creation of injection groups can be limited by only creating injection groups for which the **Selected** field is set to **YES** for these groups. To use this option, check the **Only Selected** checkbox.

### Participation Factor

The options for specifying participation factor are the same as those used on the [Add Participation Points Dialog](#).

### Delete Existing Injection Groups

When this option is checked, all existing injection groups will be deleted before inserting the new injection groups.

### How to Name the Injection Groups

The new injection groups will be named based on the options set in this section. The **Use Prefix** field determines the prefix that will be used on all injection groups. This field can be blank. The **Start at** field determines what integer to start at when counting the new injection groups. The new injection groups will all be numbered in sequential order starting at the value specified. The **Group By** property chosen will also be part of the injection group name. The **Group By** property can be identified based on **Numbers, Names**, or **Both**. The field at the bottom of this section provides an example of how the injection groups will be named based on the option settings.

### Do Insert Injection Groups

Click this button to implement the options selected and create the injection groups.

**Save to Aux**

This saves the option settings specified on the dialog to an auxiliary file.

---

## Deleting Injection Groups

---

To delete an injection group, select **Aggregations > Injection Groups** from the [Model Explorer](#) to open the [Injection Group Display](#). This display is organized into two tab pages with additional tabs and displays available under each. The Injection Groups tab lists all defined injection groups. Click the right mouse button on this display on the injection group to delete, and select **Delete** from the resulting local menu.

Alternatively, if the [Injection Group Dialog](#) is open, the injection group listed in the Name dropdown box can be deleted by clicking the **Delete** button.

---

## Injection Group Display

---

The Injection Group Records Display presents data describing each injection group in the case. This display is available by selecting **Aggregations > Injection Groups** from the [Model Explorer](#). This display is organized into two tabs with various sections available under each of the tabs. The tabs and sections provide options for displaying information about injection groups and their associated participation points in a manner in which it is most useful to the user. Injection groups can be displayed with their associated participation points or participation points can be displayed with the injection group to which they belong.

### Injection Groups Tab

This tab is organized into a top and bottom section. The top section contains the Injection Groups [Case Information Display](#) that lists all of the defined injection groups. The [local menu](#) is available by right-clicking on the display and contains options available for most case information displays as well as the option to **Show Dialog** to view the [Injection Group Dialog](#).

By default, the Injection Groups Case Information Display contains the following fields:

#### **Name**

The name of the injection group. To change the name of an injection group, simply type a new name in the corresponding cell.

#### **Number of Gens**

Identifies the number of generators contained in the injection group.

#### **% MW Gen ParFac**

Indicates the degree to which generators will contribute to the MW output of the injection group relative to loads. An injection group that has a % MW Gen ParFac value of 100% receives all of its output from generator points; an injection group that has a % MW Gen ParFac of 50% and a % MW Load ParFac of 50% receives equal MW contributions from its constituent loads and generators.

#### **Number of Loads**

Identifies the number of loads contained in the injection group.

#### **% MW Load ParFac**

Indicates the degree to which MW loads will contribute to the MW output of the injection group relative to generators. An injection group that has a % MW Load ParFac value of 100% receives all of its output from load points; an injection group that has a % MW Load ParFac of 50% and a % MW Gen ParFac of 50% receives equal MW contributions from its constituent loads and generators.

#### **% Mvar Load ParFac**

Indicates the degree to which Mvar loads will contribute to the Mvar output of the injection group relative to switched shunts. An injection group that has a % Mvar Load ParFac value of 100% receives all of its Mvar output from load points; an injection group that has a % Mvar Load ParFac of 50% and a % Mvar Shunt ParFac of 50% receives equal Mvar contributions from its constituent loads and switched shunts.

#### **Number of Shunts**

Identifies the number of switched shunts contained in the injection group.

#### **% Mvar Shunt ParFac**

Indicates the degree to which switched shunts will contribute to the Mvar output of the injection group relative to Mvar loads. An injection group that has a % Mvar Shunt ParFac value of 100% receives all of its Mvar output from switched shunt points; an injection group that has a % Mvar Shunt ParFac of 50% and a % Mvar Load ParFac of 50% receives equal Mvar contributions from its constituent loads and switched shunts.

#### **Total MW Injection**

Indicates the current total MW injection of all elements in the injection group.

#### **Total Mvar Injection**

Indicates the current total Mvar injection of all elements in the injection group.

The bottom section contains additional tabs: **Participation Points (All)**, **Generators**, **Loads**, **Switched Shunts**, and **Injection Groups**. These tabs contain tables, which are variations of the [Participation Point Records Display](#), that list all of the individual participation points for the selected injection group by element type. Participation points can be added, deleted, or modified for the selected injection group from any of the tables on these tabs. The **Participation Points (All)** table lists only the fields that are relevant for defining participation points while the specific element tables also make available the fields that are specific to that element type.

The **Generators**, **Loads**, **Switched Shunts**, and **Injection Groups** tabs have the option to **Show points that are contained by other injection group points**. This option is only relevant if an injection group contains a participation point that is another injection group. When this option is checked, the individual participation points that are contained in the injection group point will also be displayed. The **Contained by** field will list the name of the injection group that contains a particular participation point. When this option is not checked, the points displayed will be points that are contained in the selected injection group only and no additional information will be provided about the points contained inside any injection group points belonging to the selected injection group. If the **Contained by** field is empty, this means that the participation point is explicitly defined with the selected injection group and does not belong to it through an injection group point.

### **Participation Points Tab**

This tab is organized into a top and bottom section. The top section contains tabs for **Generators**, **Loads**, **Switched Shunts**, and **Injection Groups**. Tables on these tabs will list power system elements of each particular type. If the option to **Only show elements that belong to at least one injection group** is not checked, then all elements of each type in the case will be listed on one of these tabs. If this option is checked, then only those elements that belong to an injection group will be listed. The tables on these tabs are [Case Information Displays](#) for each particular type and have the full functionality of case information displays.

The bottom section is an Injection Groups Case Information Display. When an element from one of the tables in the top section of the tab is selected, this table will be populated with the list of injection groups to which the selected element belongs.

---

## Injection Group Dialog

---

The Injection Group Dialog provides information about injection groups and allows their modification. Specifically, the Injection Group Dialog lists the number of generators, loads, and switched shunts contained in the group and the percentage contribution of generators, loads, and switched shunts to the injection group's output. The Injection Group Dialog also houses the Participation Points Records display, from which points can be added and deleted from the injection group's list of participants and the various attributes of the points can be changed.

To view the Injection Group Dialog for a particular injection group, open the [Injection Group Display](#) by selecting **Aggregations > Injection Groups** from the [Model Explorer](#). Find the injection group of interest in the Injection Groups Case Information Display and right-click on it. Then select **Show Dialog** from the resulting local menu.

The Injection Group Dialog contains the following fields and controls:

### Name

Identifies the injection group whose information is currently displayed. Selecting a different name from this dropdown box will display information for another injection group.

### New, Delete

To insert a new injection group from this dialog, press **New**, and supply the name for the new injection group. To delete the injection group that is currently being shown, click **Delete**.

### Save

Changes made to injection groups through the Injection Group Dialog are not immediately saved with the power system case. This allows the option of canceling any changes without impacting the case. Click the **Save** button to store any updates with the case.

### # Gens

Displays the number of generators contained in the injection group.

### # Loads

Displays the number of loads contained in the injection group.

### # Shunts

Displays the number of switched shunts contained in the injection group.

### % MW Gen Part., % MW Load Part., % MVR Load Part., % MVR Shunt Part.

Displays the relative contributions of generators, loads, and switched shunts to the output of the injection group.

### OK, Cancel

Changes made to the injection groups through the Injection Group Dialog are not immediately saved with the power system case. This allows the option of canceling any changes without impacting the case. Click **Cancel** to close the dialog without saving the changes. Click **OK** to close the dialog and save the changes.

## Participation Points

The tab on the bottom of the Injection Group Dialog lists the points that make up the injection group. This display is called the [Participation Point Records Display](#). By right-clicking on this display, points can be added and deleted from the injection group. Properties of specific points

can be changed by entering changes directly in the table. The **Insert Points** button will open the [Add Participation Points Dialog](#) that can also be used for modifying the points.

---

**Import PTI Subsystems Dialog**


---

Injection groups can be created from PTI subsystem definition files from the [Injection Groups Case Information Display](#). Click the right mouse button on this display and select **Load > Subsystem from \*.sub file** from the resulting local menu. From the **Import Injection Groups Dialog**, select the file name for the file to import.

If there is no ambiguity in the import file, new injection groups will be created with the defined subsystem names without further prompts to the user. No ambiguity means that either participation points are defined for buses that have either load or generation but not both or that no participation points are defined and that injection group participation points will be defined based on the maximum generation for each on-line generator in the defined subsystem. If participation points are defined without ambiguity, each participation point will be assigned to either the load or generation at the bus split equally across all loads or generators at the bus.

If there is ambiguity in defined participation points, the Insert PTI Subsystems into Injection Groups Dialog will be displayed. This dialog prompts the user how participation points should be handled for buses with both load and generation or buses with no load or generation.

**Buses with Load and Generation**

This option allows the user to select how a defined [participation point](#) will be assigned if the participation point bus has both load and generation.

**Assign Participation Point to Generation**

The participation point will be assigned to generation at the bus. The participation point will be split equally across all generators, either on-line or off-line, at the bus.

**Assign Participation Point to Load**

The participation point will be assigned to load at the bus. The participation point will be split equally across all loads, either connected or not, at the bus.

**Buses with No Load or Generation**

This option allows the user to select how a defined participation point will be assigned if the participation point bus has no load or generation.

**Ignore Participation Point**

The participation point will be ignored.

**Add Equivalent Load (closed load with ID='99' and 0 MW and 0 Mvar) and Assign Participation Point to this Load**

This option adds a connected load at the participation point bus with ID='99' and 0 MW and 0 Mvar. The participation point is then assigned to this new load.

For each subsystem that is read that is found to have ambiguity in the defined participation points, the Insert PTI Subsystems into Injection Groups Dialog will be displayed unless the user selects Same Options for All Subsystems (Do not prompt again).



---

## Participation Points Overview

---

A participation point is a member of an injection group. It is a generator, load, switched shunt, or another injection group that participates in, or contributes to, the output of an injection group. Each participation point record identifies the generator, load, switched shunt, or injection group that fills this role, its participation factor, and how that participation factor is calculated. A participation point's participation factor identifies the degree to which the point will contribute to its injection group's output relative to the other points making up the group. Participation factors may be defined as having a fixed value, or they may be re-calculated with every use to stay true to how they were originally defined. For generators, participation factors may be defined as having a fixed value or being equal to the generator's MW reserve in the direction of increasing output (MAX GEN INC), MW reserve in the direction of decreasing output (MAX GEN DEC), or their maximum output (MAX GEN MW).

Each time that an injection group is used, the participation factors for all participation points in the group are normalized so that the factors of all participating participation points sum to 100%. The normalized factors dictate how much a generator, load, or switched shunt contributes to the output of an injection group. The normalization accounts for the re-calculation of the participation factors with each use of the injection groups and the exclusion of participation points because of options set with the various tools that use injection groups. Participation points can be excluded for various reasons including, but not limited to, the exclusion of generator participation points because the generator is not on AGC control or because the generator is at a minimum or maximum limit.

Participation points are added to or deleted from an injection group using the [Injection Group Dialog](#). The Injection Group Dialog houses the [Participation Point Records Display](#), which allows the addition or deletion of points and opening the [Participation Points Dialog](#) to obtain more information about the points.

---

**Participation Point Records Display**


---

The Participation Point Records Display shows information about the points that comprise a particular injection group. This display can be accessed for a particular injection group from the [Injection Group Dialog](#) or the [Injection Group Display](#).

The Participation Point Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with all other case information displays. It has a [local menu](#) from which participation points can be deleted or added to the injection group through the [Add Participation Points Dialog](#). Selecting **Insert** from the local menu will open the Add Participation Points Dialog that will allow the addition, deletion, and modification of participation points in the injection group.

The Participation Point Records Display shows the following fields by default:

**Point Type**

Every participation point is a generator (GEN), load (LOAD), switched shunt (SHUNT), or injection group (INJECTIONGROUP).

**Number**

Identifies the number of the bus to which a generator, load, or switched shunt is connected.

**Name**

Identifies the name of the bus to which a generator, load, or switched shunt is connected.

**ID**

Identifies the ID of the generator, load, or switched shunt or the name of the injection group.

**AutoCalc?**

If the value of **AutoCalc** is *YES*, the participation factor of the point is re-calculated with every use to be consistent with the way the point's participation factor was initially defined. If the value of **AutoCalc** is *NO*, the participation factor is assumed fixed at its present value.

**Initial Value**

Indicates how the participation factor of the point was originally computed. For generators, the possible values of this field are

**SPECIFIED** The participation factor was specified as a constant.

**MAX GEN INC** The participation factor was defined as the difference between the generator's maximum MW output and its present MW output.

**MAX GEN DEC** The participation factor was defined as the difference between the generator's present MW output and its minimum MW output.

**MAX GEN MW** The participation factor was defined as the maximum MW output of the generator.

For loads, the Initial Value property can assume only two possible values:

**SPECIFIED** The participation factor was specified as a constant.

**LOAD MW** The participation factor was defined as the size of the load in MW.

For switched shunts, the Initial Value property can assume four possible values:

**SPECIFIED** The participation factor was specified as a constant.

**MAX SHUNT INC** The participation factor was defined as the difference between the maximum Mvar output of the switched shunt and its present nominal Mvar output.

**MAX SHUNT DEC** The participation factor was defined as the difference between the present nominal Mvar output and the minimum Mvar output of the switched shunt.

**MAX SHUNT MVAR** The participation factor was defined as the maximum Mvar output of the switched shunt.

For injection groups, the Initial Value property can assume only one possible value:

**SPECIFIED** The participation factor was specified as a constant.

The **Initial Value** field is important if saving the injection groups to an auxiliary file and using them with another case that might have a different generation dispatch or load profile. If the **Initial Value** for a point is specified as *GEN MAX INC*, for example, and the point is loaded from an auxiliary file into another case, Simulator will re-calculate the point's participation factor to match the generator's positive MW reserve in that case.

The **Initial Value** field is also important if the **AutoCalc** field is set to *YES*, because **AutoCalc** uses the rule defined by the **Initial Value** field to recalculate the participation factor with every use. For example, if the **Initial Value** is *MAX GEN INC* and **AutoCalc** is *YES*, the point's participation factor will be updated to match the generator's MW reserve every time the point is accessed.

When the value of the **Initial Value** field is toggled, the point's participation factor, shown in the **ParFac** field, will update to match the new definition. The **Initial Value** field must equal *SPECIFIED* to be allowed to change the value of the point's **ParFac** field by typing the new value directly into the field.

If **AutoCalc** is *NO* and the injection group is not intended to be used with any other case, then the **Initial Value** field should either be ignored or set to *SPECIFIED*.

### ParFac

Indicates the participation factor of the participation point. The participation factor defines the relative contribution of the point to the total output of the injection group. The load, generation, or switched shunt change associated with each point is calculated based on the value of the participation factor, with values for points having the largest participation factors experiencing the greatest change.

The participation factor for injection group participation points indicates the relative contribution of the entire injection group to the total output of the injection group to which it belongs. The participation factors of the individual points in the injection group point further define the relative contribution of these points to the total output of the injection group. The following two injection groups provide an example of how injection group points impact the injection group to which they belong when the points are normalized for use:

#### Injection Group 1    Injection Group 2

Gen 1, ParFac = 50    Gen 5, ParFac = 40

Gen 2, ParFac = 30    Load 1, ParFac = 20

InjectionGroup 2, ParFac = 20    Load 2, ParFac = 20

Total = 100    Total = 80

When Injection Group 1 is used, the participation factors are normalized so that each element provides the following contribution:

## Injection Groups

Gen 1, Normalized ParFac =  $50/100 = 0.5$

Gen 2, Normalized ParFac =  $30/100 = 0.3$

Gen 5, Normalized ParFac =  $20/100 * 40/80 = 0.1$

Load 1, Normalized ParFac =  $20/100 * 20/80 = 0.05$

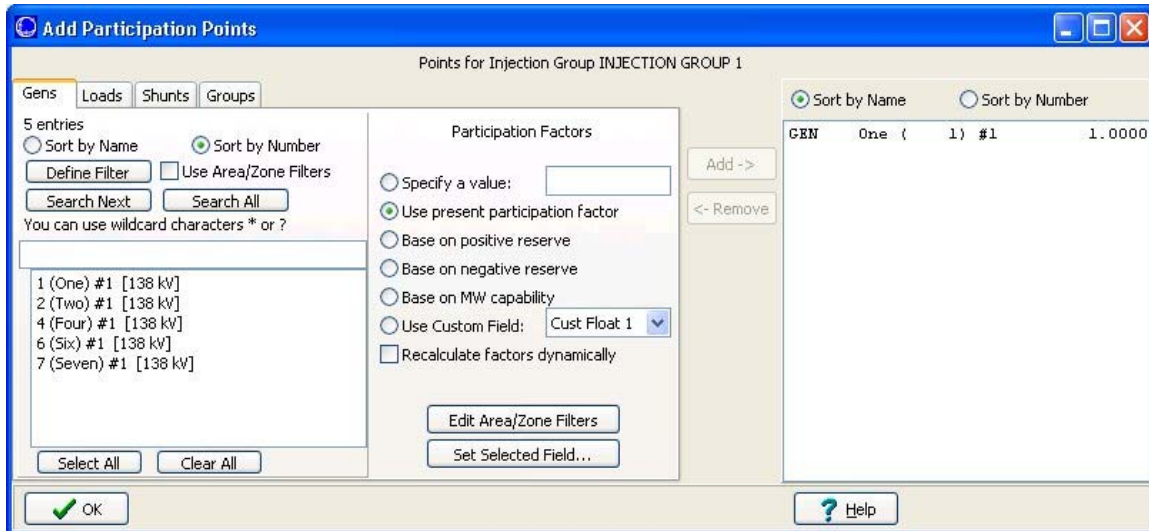
Load 2, Normalized ParFac =  $20/100 * 20/80 = 0.05$

Total Normalized = 1.0

To add points to the injection group, right-click on the Participation Point Records Display and select **Insert** from the local menu. This opens the [Add Participation Points Dialog](#).

## Add Participation Points Dialog

The Add Participation Points Dialog enables the addition of participation points to an [injection group](#) as well as the modification of existing participation points. This dialog is accessed from a [Participation Point Records Display](#) by right-clicking in the participation points list and selecting **Insert** from the resulting [local menu](#). Participation Point Records Displays are available from the [Injection Group Dialog](#) or [Injection Group Display](#).



The Add Participation Points Dialog features four tabs, one for adding generator points, one for adding load points, another for switched shunts, and a fourth for adding points from injection groups that have already been defined.

The four tabs are almost identical and contain the following controls:

### Filtering

If the **Use Area/Zone Filters** box is checked, the list box beneath it, which lists generators, loads, switched shunts or previously defined injection groups depending on the tab, will list only those elements contained in areas or zones or by owners whose [area/zone/owner filter](#) setting is *YES*. If this box is not checked, all generators, loads, and switched shunts in the case will be listed. Injection groups will not be filtered using the area/zone/owner filter.

Alternatively, a custom filter can also be defined by clicking on the **Define Filter** button. This will open the [Advanced Filter](#) dialog, which allows the customization of a filter for determining the devices to list in the display.

Once the list of devices has been set, with or without filtering, the list can be searched using the advanced search techniques in Simulator. These techniques allow the list to be sorted by name or by number and allow the use of wildcard characters. Simply choose Name or Number, and type the name or number of interest in the box. Simulator will look for and highlight the first matching device in the list. If the first device is not the one of interest, use **Search Next** to find the next device that matches the search criteria. For a comprehensive list of all objects matching the search criteria, press **Search All**.

### Element List

The box that occupies the left side of each tab lists the generators, loads, switched shunts, or injection groups (depending on which tab is active) that can be added to the injection group. Injection groups will only show up in the element list if they do not cause a circular reference by being added to the current injection group. This means that any injection group that would cause the current injection group to link back to itself will be omitted from the list. Multiple elements can be selected from each of these lists. To select several elements in a row, drag the mouse to highlight the elements to be added. Alternatively, click the first element to add, press and hold the shift key, and click the last element to add. To select elements that are not adjacent in the list, click the first element you to add and hold down the CTRL key while clicking the other elements to add.

Once an element is selected in the Element List, it is ready to be added to the injection group.

### Participation Factors

There are several options for defining the participation factors of the selected points.

For generators, a value can be specified, the generator's present participation factor (which comes from the case and is displayed in the [Generator Display](#)) can be used, the participation factor can be calculated as the difference between its present output and either the unit maximum or minimum, its maximum output capability can be used, or the value contained in a selected Custom Field can be used.

For loads, a value can be specified, the participation factor can be based on the load's size, or the value contained in a selected Custom Field can be used.

For switched shunts, a value can be specified, the value contained in a selected Custom Field can be used, or the factor can be based on positive reserve, negative reserve, or MVAR capability.

For injection groups, a value can be specified to use for every point in the injection group, the values already defined for the injection group can be used, or the value contained in a selected Custom Field can be used. These three options are available if the option to **Include Individual Group Points** is checked. When this option is checked, copies of the participation points from the selected injection groups are made and added to the current injection group with the participation factors based on the selected option. When **Include Individual Group Points** is not checked, entire injection groups are included, indicated by the Point Type of *INJECTIONGROUP* in the [Participation Point Records Display](#), in the current injection group and the only option for adding the points is to specify the value for the participation factor. When including an injection group in another injection group as the entire group, the normalization and use of the participation factors follows the discussion of ParFac found in the [Participation Point Records Display](#). Including an injection group in another injection group as the entire group is useful when the injection of the entire group needs to be changed relative to other individual elements or other injection groups.

### Recalculate Factors Dynamically

If this box is checked, the participation factors of the points being added will be automatically updated every time the points are used. Such points will then have an **AutoCalc** value of *YES*. If this box is not checked, the participation factors of the points being added will be fixed at the values defined at the time they were created.

To add the points that have been selected, click the **Add ->** button. The new points will be added to the list box on the right.

To update existing points, follow the same process for adding points and simply set the participation factors with the new values. When the **Add->** button is clicked, the existing points will be updated with the new values.

The list box that occupies the right side of each tab lists the points that already comprise the injection group. To delete specific points from the injection group, select them from this list box and click the **<-Remove** button.

To close this dialog, click **OK**.

---

## **Chapter 10: Solving and Simulating a Case**

---

This chapter describes the controls and general tools for solving a load flow in Simulator. These topics are used to customize and control the time domain and single Power Flow Solutions in the Run Mode. Additional topics also describe ways to view results and data graphically using auto-generated graphical displays.

The following material is included:

- Solution Options
- Solution and Control
- Transactions
- Charts
- Bus View Online
- Substation View Online



---

## Simulator Options

---

Simulator provides a flexible environment for simulating power system operation by offering you access to a number of customizable options. The **Simulator Options Dialog** houses pages of options that you can customize to tailor the program to your needs.

To display the Simulator Options Dialog go to the [Options](#) ribbon tab, and choose **Simulator Options** from the Case Options Ribbon Group. The Simulator Options button is also available on the [Tools](#) ribbon tab on the [Power Flow Tools](#) ribbon group.

If a timed simulation is active when you open the dialog, it is automatically paused and will not resume until you close the dialog.

There are seven categories of options for the Simulator Options dialog:

[Case Information Display Options](#)

[Environment Options](#)

[File Management Options](#)

[Limits Options](#)

[Message Log Options](#)

[Online Options](#)

[Power Flow Solution Options](#)

---

## Power Flow Solution Options

---

The Power Flow Solution Tab offers various options regarding how Simulator solves the power flow problem. There are six sub-categories on the Power Flow Solutions tab: [Common Options](#), [Advanced Options](#), [Island-Based AGC](#), [DC Options](#), [General](#), and [Storage](#). Each of these categories is shown on tabs. Many of the options in the Power Flow Solution options will be of interest only to advanced users of the package.

---

## Power Flow Solution: Common Options

---

The following options are all contained on the Common Options tab after choosing the [Power Flow Solution](#) page of the [Simulator Options](#).

**Power Flow (Inner) Loop Options** (see [Solving the Power Flow](#) for information on solution loops)

### MVA Convergence Tolerance

The MVA convergence tolerance serves as a measure for determining when the inner power flow loop of the Power Flow Solution process has reached an acceptable solution. The MVA mismatch is computed as the maximum real or reactive mismatch at any bus in the system. Usually, this value should be around 0.1 MVA. If you are having difficulty solving a particular case, it may be helpful to temporarily increase the MVA Convergence Tolerance to drive the solution closer to the actual solution, and then re-solve from this solution using the smaller MVA tolerance.

### Maximum Number of Iterations

This option defines the maximum number of iterations Simulator will perform during the Power Flow Solution process in an effort to converge to a solution. If Simulator must exceed this number of iterations, it assumes that the power flow case is not converging and will terminate the solution process. If Simulator is configured to represent non-converging power flow cases as blackouts, the screen will turn gray and the blackout warning message will appear.

### Do Only One Iteration

If checked, then Simulator will only perform one iteration of the load flow solution process when **Single Solution** is clicked, regardless of the Maximum Number of Iterations setting. This is useful when playing with a case with which you are experiencing difficulty.

**MW Control (Outer) Loop Options** (see [Solving the Power Flow](#) for information on solution loops)

### Disable Automatic Generation Control (AGC)

If checked, the enforcement of the Generation Re-dispatch to account for MW interchange constraints for all areas is disabled. By default, this option is not checked.

### Enforce Generator MW Limits

If checked, then generator minimum and maximum MW limits are enforced for all generators whose *Enforce MW Limits* field is set to true. See [Generator Information Dialog](#) for more information. Otherwise, generator MW limits are not enforced. Note that when using the economic dispatch or the Optimal Power Flow, the generator MW limits are always enforced regardless of user-settings.

**Controller (Middle) Loop Options** (see [Solving the Power Flow](#) for information on solution loops)

### Disable Checking Gen VAR Limits

If checked, the Mvar limits are ignored for all the generators in the case during a Power Flow Solution. By default, this option is not selected.

### Check Immediately

Before entering the inner power flow loop to solve the power flow equations, a decision is made about each generator in the case as to whether it will be treated as a PV or a PQ bus. It is assumed that a PV bus will maintain its setpoint voltage, while a PQ bus will maintain a constant reactive power output (for instance when the generator is at a Mvar min or max limit). Normally, the power flow equations are completely solved in an inner power flow loop using this assumption. After completing the solution, generator Mvar limits are checked to see if they have been reached (must change from PV to PQ), or if at a limit if the limit may be backed off of (must change from PQ to PV). If any generators change PQ/PV status, then the inner power flow loop is resolved.

When the **Check Immediately** option is checked, the generator Mvar limits are checked after *every* inner power flow loop iteration instead to see if generator limits have been reached, or if a generator limit may be backed off. In some situations performing this check inside the inner power flow loop can help with convergence, however performing this check does slow down the solution process. By default, this option is not selected.

#### **Disable Switched Shunt (SS) Control**

If checked, automatic control of switched shunts is disabled in all areas. By default, this option is not selected.

#### **Disable LTC Transformer Control**

If checked, automatic control of LTC transformers is disabled in all areas. By default, this option is not checked.

#### **Disable Phase Shifter Transformer Control**

If checked, then automatic control of phase shifting transformers is disabled in all areas. By default, this option is not checked.

#### **Transformer Stepping Methodology**

Choose either Coordinated Sensitivities or Self-Sensitivity Only. The default value is Coordinated Sensitivities.

When the regulated value of tap-changing or phase-shifting transformers move outside of their regulation range, then Simulator attempts to bring those transformers back inside their range. By default Simulator determines all the tap-changing and phase-shifter transformers which are out-of-range and coordinates the movement of these transformers in an attempt to bring all the regulated values back into range. Generally this results in a better convergence, however it also can be slower when a large number of transformers are out-of-range together.

Choosing Self-Sensitivity only will modify this methodology so that each transformer only looks at the sensitivity of its regulated value with respect to changing its own tap or phase. This calculation is faster, but may result in convergence problems due to transformers that interact with one another.

#### **Prevent Controller Oscillations**

Sometimes, a power flow will fail to converge because certain automatic controls such as Mvar limit enforcement at generators, transformer tap switching, and shunt switching oscillate between their control bounds. These oscillations very often are due to modeling inaccuracies. If this option is checked, Simulator will automatically detect such oscillating controls and fix them at their current value so that they no longer oscillate. You may find this option helpful if you feel that the modeling of automatic controls in your system is inaccurate.

#### **Maximum Number Controller Loop Iterations**

As part of the solution process, the outer loop of the solution algorithm is a check of any necessary controller changes due to changes in controlled values from the last iteration of the Newton-Raphson load flow solution. The maximum number of loops through the control change algorithm can be set here. This is not the same as the Maximum Number of Iterations, which applies to the actual Newton-Raphson inner loop algorithm, which solves that actual power flow.

---

## Power Flow Solution: Advanced Options

---

The following options are all contained on the Advanced Options tab after choosing the [Power Flow Solution](#) page of the [Simulator Options](#).

### **Dynamically add/remove slack buses as topology is changed (Allow Multiple Islands)**

If checked, multiple islands are allowed during the solution by Simulator dynamically choosing a new slack bus for the island. If there are no in-service generators in the new island, the island cannot be solved and will be isolated and ignored during the load flow solution.

### **Post Power Flow Solution Actions**

Clicking this option will open the [Post Power Flow Solution Actions dialog](#), where the user can specify a list of actions to be executed at the end of every Full AC power flow solution.

### **Power Flow (Inner) Loop Options (see [Solving the Power Flow](#) for information on solution loops)**

#### **Disable Power Flow Optimal Multiplier**

If checked, the Newton solution process will not use the optimal multiplier. The optimal multiplier is a mathematically calculated step size for the Newton's Method iteration that prevents the mismatch equations from increasing between iterations. A small value indicates that moving in the direction of the Newton step will increase the mismatch equations. When the optimal multiplier becomes too small, this is an indication that the power flow is not going to solve successfully and Simulator will stop instead of creating extremely large mismatches. If this happens at a point where the solution is not within the allowed tolerance for the Newton process, the Newton process will result in a failed convergence to a valid solution.

#### **Initialize From Flat Start Values**

When checked, each Power Flow Solution is started assuming that all voltage magnitudes and generator set point voltages are unity and all angles are zero. By default, this option is not selected. Some power flow problems can be very difficult to solve from flat start assumptions. Therefore, use this option sparingly.

#### **Minimum Per Unit Voltage for Constant Current Loads**

This option is used to model the impact that falling voltage has on loads throughout the system. The default value is 0.5.

For constant current loads with a terminal bus below this per unit voltage, the value of the load will fall off using a sine function so that the load is zero at zero terminal voltage, and so that the derivative is continuous at the minimum per unit voltage for constant current loads.

#### **Minimum Per Unit Voltage for Constant Power Loads**

This option is used to model the impact that falling voltage has on loads throughout the system. The default value is 0.7.

For constant power loads with a terminal bus below this per unit voltage, the value of the load will fall off using a cosine function so that the load is zero at zero terminal voltage, and so that the derivative is continuous at the minimum per unit voltage for constant current loads.

### Control (Middle) Loop Options (see [Solving the Power Flow](#) for information on solution loops)

#### Disable Treating Continuous Switched Shunts (SSs) as PV Buses

Continuous switched shunts are normally treated the same as a generator bus inside the inner power flow loop (they are treated as buses with fixed power and voltage). Checking this option will cause the continuous switched shunt to be treated as a constant impedance in the inner power flow loop with all switch occurring in conjunction with other switched shunts in the voltage control loop instead.

#### Disable Balancing of Parallel LTC Taps

Simulator has the capability to attempt to balance tap positions of parallel transformers, in an attempt to avoid parallel transformers from going to opposite tap settings, inducing loop flow through the parallel transformers. Checking this option disables the automatic balancing of parallel transformers. The only transformers that will be balanced are those in parallel between the same terminal buses, or those in parallel between terminal buses that are connected with zero impedance branches.

#### Model Phase Shifters as Discrete Controls

If checked, then phase shifters will switch tap positions discretely based on the tap step size of the phase shifting transformer. By default, this option is not checked, which means the phase shifters will switch continuously, independent of the tap step size.

#### Minimum Sensitivity for LTC Control

This option specifies the minimum-voltage-to-tap sensitivity for LTC transformers. All transformers having an absolute value of voltage-to-tap sensitivity below this value are automatically disabled from automatic control. This prevents Simulator from changing transformer taps that have little effect on their controlling voltage. The [Transformer AVR Dialog](#) shows the voltage-to-tap sensitivity for each voltage-controlling transformer.

#### Disable Angle Rotation Processing

At the end of a power flow solution, by Simulator looks at the bus voltage angles in the case to see if any are near +/- 180 degrees. If the angles fall outside of +/- 160 degrees, then all angles in the island will be rotated by the same amount so that the angle range in the islands is equally spaced around zero degrees. Check this option to disable this feature.

#### Sharing of generator Vars across groups of buses during remote regulation

When several buses have generators that control the voltage at single bus, this option determines the method used to determine each buses "share" of the MVar support. There are three options

1. Allocate across buses using the user-specified remote regulation percentages. This option is what is used by default and most closely matches the sharing seen in RAW files.
2. Allocate so all generators are at same relative point in their [min .. max] var range. This option most closely matches the sharing seen in a few EMS solutions PowerWorld has seen.
3. Allocate across buses using the SUM OF user-specified remote regulation percentages. This option most closely matches the sharing seen in EPC files.

It should be noted, generators at the same bus always allocate vars so they are at the same relative point in their [min..max] MVar range. How vars are allocated to generators at the same bus has no effect outside of that bus so options are not given for this sharing.

### **Include Loss Penalty Factors in ED**

If checked, the economic dispatch calculation will consider losses in determining the most economic generation dispatch. Otherwise, the generation dispatch calculation will disregard system losses.

### **Enforce Convex Cost Curves in ED**

The economic dispatch algorithm attempts to set the output of all generators that are set to be automatically controlled so that the system's load, losses, and interchange are met as economically as possible. The algorithm is guaranteed to reach a unique solution only when all generator cost curves, which model the variation of the cost of operating a unit with its output, are convex. If this option is checked, Simulator will identify units whose operating point is outside the convex portion of the cost curve and set them off automatic control.



---

## Post Power Flow Solution Actions Dialog

---

The Post Power Flow Solution Actions dialog describes a list of actions that are executed at the end of every Full AC power flow solution, which means they are not performed for DC solutions. Normally, these actions would all have Model criteria specified.

To open this dialog, select **Simulator Options** from the **Case Options** ribbon group on the [Options](#) ribbon tab. Next, go to the **Power Flow Solution** category. Then click on the **Advanced Options**. Finally click the button labeled **Define Post Power Flow Solution Actions**.

Check the **Do Not Used Post Power Flow Solution Action List** check-box to disable using any actions defined in the list.

The action list display identifies the actions that comprise the post power flow solution action list. Actions can be inserted or deleted by using the local menu on the dialog. Right-click on the display and select **Insert** or **Delete**. Actions are inserted via the [Contingency Element Dialog](#).

The action display is a type of [Case Information Display](#) and thus shares many common to all other case information displays.

The Action List Display always contains the following fields:

### Actions

This shows a string which describes the action. You may customize the format of the string that describes the actions by right-clicking on the Action List Display and choosing **Display Descriptions By**, and then choosing either Name, Num, Name/Num, PW File Format by Numbers, PW File Format by Name/kV or PTI File Format.

### Model Criteria

Simulator allows you to define Model Criteria, which consist of both [Model Conditions](#) and [Model Filters](#). These specify a criteria under which a contingency action would occur. For example, you could specify that a generation outage only occur if the pre-contingency flow on a line is higher than a specified amount. Normally, no Model Criteria will be specified, and this field will be blank. Also, note that Model Criteria can be overridden by the Model Condition and Filter option on the [Contingency Options Tab](#). You can open a dialog to define Model Filters or Conditions by right-clicking on the Action List Display and choosing **Define Model Criteria**.

### Status

Three options are available for this field

- **CHECK** : The action will be executed only if the Model Criteria is true. It will also be executed if no model criteria is specified. This is the default setting
- **ALWAYS**: The action will always be executed, regardless of the Model Criteria.
- **NEVER**: The action will never be executed, regardless of the Model Criteria.
- **POSTCHECK**: This action is checked **AFTER** the other Check and Always actions have been performed and the load flow solution solved. If the criteria specified for the Postcheck action are met in the resulting load flow solution, then this action is taken and the load flow is again resolved. This will recursively occur for all Postcheck actions until either all postcheck actions have been taken, or the criteria for all remaining postcheck actions have not been met.

Note that the Never action allows you to disable a particular action without deleting it.

### Comment

A user-specified comment string that can be associated with this action. While this comment is not used by Simulator in any way, these comments can be saved to be loaded at a later time. This is provided for the user to add comments regarding the action. For example, for an action with a Model Criteria you may could add a sentence explaining why the action is only performed under the specified criteria.

---

## Power Flow Solution: Island-Based AGC

---

The following options are all contained on the Island-Based AGC tab after choosing the [Power Flow Solution](#) page of the [Simulator Options](#).

The Island Based AGC options allow the user to choose to dispatch generation by island instead of by area or super area. The options available for dispatch here are:

### **Disable (Use the Area and Super Area Dispatch Settings)**

When this option is selected, Island Based AGC is being used, and generators are dispatched according to the usual area or super area generation dispatch.

### **Use Participation Factors of individual generators**

When selected, the island-based AGC is used, based on the individual participation factors of each generator within the island. Area and super area ACE requirements will be ignored, and all generators in the island will be dispatched to serve load, losses, and any DC transfers to other islands.

### **Calculate Participation Factors from Area Make Up Power Values**

The AGC will dispatch generation by use area make up power values, instead of by individual participation factor. What this means is that each area will be assigned a make up power value, similar to assigning a participation factor to a generator. Based on the make up power value of each area in the island, the amount of the generation dispatch needed will be divided amongst each area based on its value. Higher value will account for more of the generation dispatch than areas with smaller values. The within each area, the generation dispatch is handled on an individual generator participation factor basis, where each generator will account for a portion of the dispatch that was assigned to its area.

To specify the make up power values for each area, click on the **Specify Are Make Up Power Values** button. Note that these settings are the same as those used by [Contingency Analysis Options Make Up Power](#), therefore changing these values will affect the contingency analysis.

For example, three areas have participation factors of 2, 1 and 1, respectively. If the total generation redispatch in the island is 100 MW, then area 1 will account for  $2 / (2+1+1)$ , or 50%, of the total. Therefore area 1 is expected to redispatch by 50 MW. If area 1 then has two generators with participation factors of 4 and 1, they will account for the 50 MW by picking up  $4 / (4+1)$  and  $1 / (4+1)$ , or 80% and 20%, respectively, of the 50 MW needed from the area.

### **Dispatch using an Injection Group (Loads and Generators will respond)**

Checking this option will allow for the island dispatch to be covered by change in generation and/or load defined in an [injection group](#).

Additional options are available when dispatching based on an injection group:

#### **Allow only AGC Units to Vary**

If this option is checked, only units whose AGC status is turned on will be allowed to participate in the injection group dispatch.

#### **Enforce unit MW limits**

If checked, then each generator's defined MW limits will be strictly adhered to during the redispatch.

#### **Do not allow negative loads**

When checked, loads included in the injection group are not allowed to drop below zero MW or MVAR load demand.

**How should reactive power load change as real power load is ramped?**

You can choose to keep the ratio of real and reactive power constant for each load that is included in the injection group, or you can specify a constant power factor that the MVAR value will be determined from when the MW value is changed.

---

## Power Flow Solution: DC Options

---

The following options are all contained on the DC Options tab after choosing the [Power Flow Solution](#) page of the [Simulator Options](#).

### Use DC Approximation in Power Flow

When this box is checked, Simulator will solve the load flow using the DC Approximation method. When not checked, Simulator performs the modified Full Newton AC load flow algorithm. Note that once a case, especially a large one, has been solved using a DC approximation, it tends to be quite difficult to revert back to the Full Newton AC load flow from a solved DC approximation.

### DC Power Flow Model

Note: These choices also affect the calculation of sensitivities on the [PTDF](#), [LODF](#), and [TLR](#) dialogs and [ATC Tool](#) calculations.

Generally, the *only* inputs that affects the DC Power Flow equations are line impedances and line status (open/closed). However there are three possible modeling choices (for which there is not general agreement in the industry) that will affect the "DC equations" in small ways. These choices are made here on the DC Power Flow Model options

#### Ignore Series Resistance (r) or Ignore Series Conductance (g)

The series term of a transmission line consists of an r and x value which represent a complex number impedance. In the DC power flow approximation, the value that used is the imaginary part of the inverse of impedance (called admittance)

$$g + jb = 1 / (r + jx) = r / (r^2 + x^2) - j x / (r^2 + x^2)$$

Thus

$$g = r / (r^2 + x^2)$$

$$b = -x / (r^2 + x^2) \quad *** \text{ This term is used in the DC power flow equations}$$

The only term that is used in the DC power flow equations is the b term. Some say that a DC power flow means that r = 0 which means that b = -1/x and g = 0. Others say that a DC power flow only means assuming that g = 0, which means that b = -x / (r^2 + x^2). There is not a good consensus in the industry as to what is correct regarding this, so Simulator offers an option as to whether you ignore r or ignore g. The default option in PowerWorld is to ignore the resistance.

#### Ignore Transformer Impedance Correction Tables and Ignore Phase Shift Angle Effects

These are both ignored by default.

When a transformer impedance correction table is specified for a transformer, then the series impedance of the transformer will vary by a multiplier with the tap or phase of the transformer. The tap ratio is not generally relevant to a DC power flow, however the phase-shifter angle could vary when using the DC approximation. This option specifies whether to ignore the impact of this impedance correction. For phase shifters the multiplier is normally between 1.00 and 1.50 and thus tends to *increase* the impedances as you move away from zero degrees.

As a branches phase shift varies, the DC power flow equations could be modified to take this into account. This effect results in the DC power flow equations for which the "b" term above is multiplied by the cosine of the phase angle across the line. This effect tends to *decrease* the impedance as you move away from zero degrees.

Because the impedance correction tends to increase impedance and the phase effect tends to decrease impedance it is reasonable to ignore both of these, and this is done by default. The other issue with not ignoring these is that both create a situation where the equations for the DC approximation are dependent on the system state. In other words, the matrix equations become a function of the phase-shift angle. This results in a situation in which the matrices must be recalculated and re-inverted each time a phase angle is moved. This removes some of the advantages of the DC approximation. This can be especially problematic when using the OPF or SCOPF.

#### **Compensate for Losses by Adjusting the Load**

Traditionally a DC load flow is treated as lossless. However, you can approximate the loss in the load flow by artificially adjusting the load in the case to include estimated losses. To do so, click on the **DC Loss Setup** button to open the [DC Power Flow Loss Setup](#) dialog for setting the DC Loss Multipliers.

#### **Compensate for Reactive Power Flows by Adjusting the Branch Limits ...**

One issue with DC load flow is how to treat the inclusion of reactive power flows. In a standard DC load flow, the reactive flows are typically ignored, and the MW flow of a branch is compared to its original MVA branch limit. However, Simulator also gives you the option to instead solve the DC load flow and compute approximate MVAR flows by assuming the voltages in the system remain constant when the DC load flow is solved.

#### **Compensate for Dispatch Sensitivities with User-Specified Values**

This option allows for the bus MW loss sensitivities to be used in the OPF and ED dispatch algorithms, if the type of loss sensitivity on the [General](#) tab is set to User-Specified.

---

## DC Power Flow Loss Setup

---

To open the DC Power Flow Loss Setup dialog, select **Simulator Options** from the **Case Options** ribbon group on the [Options](#) ribbon tab. On the Power Flow Solution page, click on the tab labeled **DC Options**. Next, click the button labeled **DC Loss Setup**. This button will only be enabled when you check the box to **Use DC Approximation in Power Flow/OPF/SCOPF**.

The DC Power Flow Loss Setup dialog gives you a location to apply approximate losses during a DC load flow solution. The losses can be approximated by scaling the loads in the case to include an approximation of losses. Loss multiplication factors can be applied individually by bus, or as a group by area or by zone. Note that loss multipliers by area or by zone are just quick ways for setting the bus multiplication factors for all buses in the group. You will see the value reflected for all buses in the Buses page. If you wish to apply the same multiplication factor to the entire case, you can simply set the Case DC Loss Multiplier at the bottom of the dialog. This will automatically set all buses in the case to have the same DC Loss Multiplier specified.

Once the DC Loss Multipliers have been set, click OK to save the multipliers.

---

## Power Flow Solution: General

---

The following options are all contained on the General tab after choosing the [Power Flow Solution](#) page of the [Simulator Options](#).

### Assumed MVA Per-Unit Base

This option allows the user to specify the MVA base to be used for the entire case. By default, this value is set to 100 MVA.

### Bus Loss Sensitivity Function

Bus loss sensitivities indicate how island or area losses change with power injection at the bus. Here you may choose to forego the calculation of bus loss sensitivities or to base them on island losses or area losses. If the case consists of only one island, which, by definition, corresponds to the entire system, then the bus loss sensitivities are measured with respect to total system losses. If the bus sensitivities are set to User-Specified, the sensitivities will remain at their last calculated values, according to the loss function type previously specified when the [loss sensitivities](#) were calculated.

### Monitor/Enforce Contingent Interface Elements

This global location allows you to determine how contingency elements in an interface should be treated in Simulator. You can choose to never include the impact of contingent elements on interface flow, to only include contingent element impacts in the standard power flow or optimal power flow routines, or in all solution routines including contingency analysis and security constrained OPF.

It is not uncommon to ignore the impact of contingent elements when using the contingency analysis or security constrained OPF tools, as they are already processing lists of contingencies and evaluating flows on interfaces. Ignoring contingent elements within interface definitions allows for a determination of the impact of other contingencies on the flows of the non-contingent elements forming the interface, without impact from additional contingent element considerations.

### Power Units for Display

Allows switching between displaying power values in Mega- units or Kilo- units.



---

**Power Flow Solution: Storage**

---

The following options are all contained on the Storage tab after choosing the [Power Flow Solution](#) page of the [Simulator Options](#).

Simulator offers the ability to restore either the last power flow solution state or the state of the system immediately before the last solution attempt. If your system has insufficient memory and you are working with large systems, you may wish to disable one or both of these options.

**Disable "Restore last solution"**

Restoring the last solution will undo any changes made to the data that were made after the last successful solution and return the case to the last valid solution. For large cases, the amount of memory required to store the last solution can be significant. If this option is checked, Simulator will not store this information in memory, and the last solution cannot be restored if a solution fails.

**Disable "Restore state before failed solution attempt"**

Restoring the state before failed solution attempt will undo only the attempted solution, but will retain any changes to data that were made before the solution process. This allows the user to return to the point just before the solution in order to add or remove changes in an effort to obtain a valid solution. For large cases, the amount of memory required to store the state before a solution attempt can be significant. If checked, Simulator will not store the state information in memory, and the state before the failed solution cannot be restored.

---

## Message Log Options

---

### Show Log

If checked, the message log is displayed. The message log shows detailed results of each Power Flow Solution. Usually this log is NOT shown. However, if you are running into problems with a simulation case, it can prove useful for debugging the case.

### Show Time Stamps

Checking this option will display the date and time at which each message is posted in the message log.

### In log Messages, Identify buses by

This option allows the user to specify how to identify the buses in log messages. The options are by numbers, by names, or by both numbers and names.

### Include Nominal Voltages in Log

If checked, this option will make the buses to be displayed with their nominal voltages after their name.

### Suppress the following messages in the log

Checking the check-boxes in this option will remove the corresponding message writing to the log, thus speeding up the computation process. The boxes to the right of the messages indicate the color with which the messages will be displayed in the log.

---

## Environment Options

---

The Environment page provides you control over a number of display and simulation options. The first section of the page contains a list of check boxes that you can use to designate the content of the oneline displays.

These options include:

### Show Log

If checked, the message log is displayed. The message log shows detailed results of each Power Flow Solution. Usually this log is NOT shown. However, if you are running into problems with a simulation case, it can prove useful for debugging the case.

### Disable Showing Blackouts

You can dramatize a power flow case's failure to converge by representing it as a blackout. The background of the oneline diagram will become a dark shade of gray, and a message box will appear to announce that the system has experienced a blackout. To disable this behavior, select the Disable Showing Blackouts box. Representing the failure to converge as a blackout can be very helpful for presentation purposes. Very often, the power flow's failure to converge can be traced to the system's inability to serve the load demand, a situation that requires that load be "blackened out," or shed, to restore the system to a viable operating state. Thus, displaying the failure to converge as a blackout has physical significance.

### Disable AGC When Manually Changing Generator MW

When this option is checked, changing the MW output of a generator manually will automatically remove a generator from [Automatic Generation Control](#). If you wish for generators to maintain their automatic generation control settings following a manual change of MW output, you must uncheck this option.

### Open Associated Oneline Diagrams

By default, Simulator has always attempted to open any diagrams that were open with a case file when the case file was last closed. However, now you may uncheck this option and Simulator will not attempt to open any oneline diagrams when a case is loaded.

### Auto Open Bus Records if No Oneline

If a case is opened in Simulator which does not have an associated oneline diagram, then Simulator will automatically open the [Bus Records](#) case information display if this option is checked.

### Allow to Prompt Dialog to Create Oneline Diagram

If a case is opened in Simulator which does not have an associated oneline diagram, then Simulator will prompt a message dialog that allows the user to create a blank oneline diagram.

### Auto Set Case as Base Case for Difference Flows when Loaded

This option allows the case to be set as a base case automatically after being loaded. This can be done always or never, or only if the base case has not been set before opening the case.

### Recently Used File List Entries

The maximum number of file names and locations stored in the History List of the File menu.

### Undo Memory Limit Per Oneline

The approximate maximum number of Megabytes allowed to be used in storing actions carried out when editing a oneline diagram, which can be undone at any time.

### **Custom Colors**

This option allows the user to edit the custom colors used throughout all Simulator color dialogs.

### **Clock Style**

The clock serves as a timer for timed simulations by showing the current time, the start time, and the end time of the simulation. You can choose to hide the simulation clock by specifying a clock style of *None*. Otherwise, to display the clock in its own window, choose *Dialog*, and to display the clock on the program's status bar, select *Status Bar*.

### **Measurement System**

This option allows the user to choose English (Imperial) or Metric (SI) units for system measurements. By default, this option is set to English units.

---

## Online Options

---

These options are available on the Online tab of the [PowerWorld Simulator Options](#) dialog. It is important to note that the options on this dialog apply to *all* online diagrams. There are also a large number of options which apply only to an individual online diagram. The diagram-specific options are specified on the [Online Display Options Dialog](#).

### Do Not Solve While Animating (Display Only)

If checked, Simulator only displays the case; it does not solve the power flow equations. System flows are determined by the initial values in the case file. This option should be checked if you simply want to use Simulator to visualize a case that has already been solved. The advantage of the display-only mode is that animation is significantly faster, particularly for large cases. The drawback to the display-only mode is that the power flow equations are not automatically solved at each time step; you must explicitly call for a Power Flow Solution using either the **Single Solution** button in the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab or choose one of the solution options under the **Solve** menu in the [Power Flow Tools](#) ribbon group.

### Play/Animation Solution Method

This option is available only for OPF releases of Simulator. Use it to indicate whether Simulator's repetitive power flow should perform a normal power flow solution or an optimal power flow (OPF) solution.

### Solution Animation

Check Auto Start Solution Animation to automatically start the animation after loading the case.

Check Auto Solve On Load to automatically solve the power flow solution when the case is loaded.

### Visualizing out-of-service elements

These three options allow you to choose how objects on the diagram should appear when they are representing a power system device that is currently "open" or "out-of-service." The three options are Blink, Use dashed lines, and Draw and X through off-line generators. The first two options apply to any online object, while the third option is specific to generator objects only.

### Main Online File

This option is used to identify the primary online diagram to use with the case. The main online is the file that is displayed when you first open the case. The dropdown box lists all the online files that reside in the same directory as the case. Select one of these files, or enter the full path of the online you want to use if it does not appear in the dropdown box.

### Use Default Online File

You can command Simulator to open a particular online diagram file if it cannot find a online diagram file for the case you are trying to open. For example, there is no online diagram associated with a PSS/E raw data file when you first read it into Simulator. However, if your application is such that you will always use the same online file whenever you open a PSS/E raw file, check the **Use Default Online File** to have Simulator open the online identified in the **Default Online File** box whenever it encounters a case that has no associated online. The default online must exist in the same directory as the case you are trying to open.

### **Show Online Hints**

If checked, pop-up hints will appear when you drag the mouse over an object in the oneline. These give information about the object; for example, for a generator the pop-up hint displays the bus number, generator ID, the MW output and the MVAR output. You may also customize the pop-up hints for each kind of display object by choosing [Options > Custom Hint Values](#) from the menu.

### **Show X,Y Coordinates**

If checked, the (x,y) location of the cursor is monitored in the status bar at the bottom of the screen. The (x,y) location of the cursor is only shown in Edit Mode. By default, this option is selected.

### **Save Contour Image with Online File**

If checked, and if a contour is being displayed on the oneline diagram, Simulator will store the contour with the oneline diagram when you save the case or save the oneline. It will store the contour as a bitmap.

### **Display Unlinked Elements in Run Mode**

Typically, unlinked graphical objects are not visible in Run Mode. Checking this option will allow the display of unlinked elements during Run Mode.

### **Enable Mouse Wheel Zooming**

This option allow the use of the mouse wheel to zoom in and out in the currently selected oneline diagram.

### **Minimum Screen Font Size**

The minimum font size at which text is visible on the screen. When text is rendered to the screen, any text which is smaller than this font size will not be rendered. This is useful when zooming out on a oneline diagram where a lot of text might become cluttered or hard to read on the screen.

### **Minimum Print/Copy Font Size**

The minimum font size at which text can be printed or copied. This is useful if the application or printer you are sending to can or cannot display smaller fonts. When text is printed, any text which is smaller than this font size will not be printed. Normally the Minimum Print/Copy Font Size is smaller than the Minimum Screen Font Size because the resolution of the printed document is much higher than the screen resolution.

### **Transformer Symbol**

Since transformer representation varies in different countries, this option allows the user to represent transformers as coils or circles. By default, transformers are represented as coils.

### **Save Onelines when Saving Case**

By default, Simulator always saves any oneline diagrams that are open when the user chooses to save the case (pwb) file. This option allows you to choose to be prompted to save oneline diagrams when a case file is saved, or to never save onelines when the case file is saved.

### **Edit Online Browsing Path**

This option applies when you have [Online Links](#) included on a oneline diagram. Rather than specify the full path and name of a oneline diagram as a oneline link, you can specify the file name only. When the link is clicked in Run Mode, Simulator will check all directories listed here, in order, to try and find the oneline file name stored with the link. This browsing path

is also used when looking up [Shapefile Database Records](#) and when finding the substation layout, URL, or command specified on the [Substation Information Dialog](#).

---

## Case Information Display Options

---

PowerWorld uses numerous [case information displays](#) to show [javascript:RelatedTopic1.Click\(\)](#) in tabular format. The options presented on this page of the PowerWorld Simulator Options Dialog control some of the general features of the case information displays.

### Enterable Field Color

Fields whose values can be directly entered on the case information displays are colored navy blue by default. Click on the color field to change the color of enterable fields, or click the **Change** button.

### Toggleable Field Color

Fields whose values can be toggled (changed) by left-clicking on them are colored green by default. Click on the color field to change the color of toggleable fields, or click the **Change** button.

### At or Exceeding Limit Color

Fields whose values are at or exceeding a limit, are colored red by default. Click on the color field to change the color of such fields, or click the **Change** button.

### Normal Field

Fields that cannot be modified directly from the case information display are colored black by default. Click on the color field to change the color of such fields, or click the **Change** button.

### Special External Field

Fields that can be modified through an auxiliary file or by pasting data into a sheet from a spreadsheet, but cannot be changed in a case information display, are colored black by default. Click on the color field to change the color of such fields, or click the **Change** button.

### Field not presently used

Fields whose values are overridden by other instances of the program are colored gray by default. Click on the color field to change the color of such fields, or click the **Change** button.

### Background

Background of cells is colored white by default. Click on the color field to change the color of the background, or click the **Change** button.

### Heading Background

Background of column and row headings is colored light gray by default. Click on the color field to change the color of the heading background, or click the **Change** button.

### Data Fill Background Color

Background of selected cells when propagating values is colored yellow by default. Click on the color field to change the color of the heading background, or click the **Change** button.

### Set Case Info Factory Default Colors

Clicking the **Dark Colors** button will reset the field and background colors to the defaults mentioned above, which comprise dark colors for fields, and light colors for backgrounds. Clicking the **Light Colors** button will set field and background colors to a specified set of light colors for fields and dark colors for backgrounds.

### Save as Auxiliary File Data Format



This option allows the user to decide if the auxiliary files that can be saved from case information displays should be space delimited files or comma delimited files.

### **Key Fields to Use in Subdata Sections**

This option allows the user to decide whether to use primary or secondary key fields or labels for the objects saved in the subdata section of an auxiliary file. In addition to being applied to the subdata section of auxiliary files, this option is also used in some special cases to dictate how elements are identified. The [Labels](#) topic contains additional information about this.

### **View/Modify Default Font**

Clicking this button brings up a font dialog from which you can choose the font in which case information displays should show their data. Selecting a new font, font size, style, or color and pressing OK will change the default font, so that all case information displays will then employ a font having the selected properties.

### **Default Row Height**

This option sets the height of the rows in the case information displays. This field may need to be changed depending on the screen size and font size of the computer. By default, the height is set to 16.

### **Show Grid Lines**

Grid lines are shown by default in case information displays. Unchecking this box will remove the grid lines from the displays.

### **Column Headings**

This option allows you to choose whether the column headings of the case information displays are the normal column headings or are the variable names of the type of data stored in each column. Variable names are the strings used when writing data out to an [Auxiliary File](#). Change the headings to variable names can be helpful when looking at or creating auxiliary files.

### **Use Word Wrap**

Check this box to wrap the heading column text, breaking it in several rows to fit the current column width. If unchecked, the column heading text will be displayed in only one row.

### **Highlight Key and Required Field Column Headings**

If this option is checked, the key fields needed for identifying objects will be highlighted in the case information display. These fields are important if you are planning on pasting data back into Simulator from Excel or reading data in from an auxiliary file. If the necessary key fields are not present in the Excel paste of auxiliary file, the data in those formats cannot be processed by Simulator.

Similarly, the Required Field Column Headings can also be highlighted in the case information display. In order for Simulator to create new objects by reading them from an auxiliary file or and Excel paste, the required fields must be present in the pasted data or the object cannot be created.

### **Show Header Hints**

This option allows the hints for the current field to be displayed when moving the cursor over the column heading.

### **Copy/Send Options**

Typically when you copy information to the clipboard or send data to Excel from a case information display, the first two rows of the copied information contain the type of object

the data represents (object name) and the column headings for each column of data. These rows are necessary if you intend to [paste the information back into Simulator](#), but they are unnecessary if you are only exporting data to another program with no intention of pasting the information back into Simulator. Thus, these two options allow you to choose which of the two rows, if either, you wish to have copied along with the actual data from a case information display when pasting in another application.

#### **Disable Auto Refresh**

This option prevents Simulator from automatically updating the contents of open case information displays with each solution. If this option is not checked, the data in all open case information displays will be updated automatically to reflect the system state calculated from each Power Flow Solution.

#### **Set Factory Defaults**

Clicking on this button will reset the options to their defaults.

---

## Limits Options

---

These options can be found on the [PowerWorld Simulator Options](#) dialog, and are used for options specific to limit enforcement or identification. Some of these settings are also available in other locations.

### Enforce Generator MW Limits

If checked, generator MW limits are enforced.

### Enforce Generator Ramp Limits

If checked, generator MW ramp limits are enforced.

### Automatically Open Overheated Lines During Simulation

If checked, overheated lines automatically open during simulation.

### Highlight Field of Objects with Limit Violations

If checked, the fields for objects on the diagram that correspond to device parameters with range limits that are violated will be colored according to the **High Limit Color** and **Low Limit Color** fields. The colors of these two fields can be defined by the user. This highlighting applies only to the following fields:

- Bus voltage
- Interface MW
- Line MW, Mvar, MVA and Amp
- Generator MW and Mvar
- Load MW
- Switched Shunt Mvar
- Transformer Phase and Tap Ratio

If other highlighting is desired see the more flexible [Dynamic Formatting](#) feature. Using Dynamic Formatting, much more complex highlighting of any field or object can be achieved.

---

## File Management Options

---

There are two sub-categories on the File Management Options tab: PowerWorld Files, and EPC and RAW Files.

### PowerWorld Files

#### Automatic Loading of Auxiliary File

A default auxiliary file can be loaded when the present case is opened by checking the **Automatically load an auxiliary file when the present case is opened** check box. The auxiliary file given under **Auxiliary File** will then be loaded with the present case. The full path needs to be included with the file name.

#### Automatic Loading of Auxiliary File with ANY Case

A default auxiliary file can be loaded when each case is opened by checking the **Automatically load an auxiliary file when ANY case is opened** check box. The auxiliary file given under **Auxiliary File** will then be loaded with each case. The full path needs to be included with the file name.

#### Automatic Archiving of PWB Files

This option allows you to effectively make backup copies of your working case every time you save the pwb file. For example, assume you have a case named Test Case.pwb. With the Automatic Archiving turned on, saving the case will first create a copy of the original file and rename it Test Case\_1.pwb. The character used as a delimiter can be chosen optionally. The case with any changes you have just made will then be saved as the new Test Case.pwb file. Each time you save the case, the latest version is named Test Case.pwb, the last Test Case.pwb is renamed with the delimiter and 1 appended, and all other archived versions will be renamed with their number incremented by 1. The number of archive versions to maintain can also be chosen by setting the maximum number of archive files property. The automatically archived cases will be saved in the same directory from which the last case was opened.

#### Autosave Every ... Minutes

This option allows the user to specify how often (in minutes) the case is automatically saved. A value of zero means that this option will not be used.

#### Save Unlinked Elements of contingency, interface and injection group records in the PWB file

Checking this box allows saving any unlinked contingency, interface, and injection group records in the PWB case. The unlinked elements are only created when read from an auxiliary file containing unlinked records.

### EPC and RAW Files

#### Display GE Additional Data in Case Information Displays

A large amount of data which is not relevant to the Simulator power flow solution is contained in the GE EPC file. This data is now all read and stored in Simulator and Simulator's PWB file format. By default columns which represent this extra data are not available when choosing [Display/Column Options](#) on a [Case Information Display](#). If you would like these fields to be available, check this box.

#### Load Additional Data from ECF

Older versions of Simulator did not read all the extra data from the GE EPC file and instead stored it in files with the ECF extension. If you would like to read in one of these old ECF files, click this button.

#### **Remove Additional Data from Case**

Because all of the extra data in the EPC can take up a lot of space (approximately doubles the PWB file size and increases the memory foot-print of Simulator itself too), you may want to just delete all this information. Click this button to permanently remove this data from your case.

#### **GE EPC Motor Data Table**

Part of the extra data read from the EPC file is Motor Data which does not map to an object used inside PowerWorld. To see this information, click on this button.

#### **PTI RAW Transactions due to tie-line loads**

Loads may be assigned to a different area than the load's terminal bus. PTI offers an option that allows the user to ignore this assignment when calculating the tie-line flow for an area.

(AREA INT CODE = 1 FOR LINES ONLY)

(AREA INT CODE = 2 FOR LINES AND LOADS)

Simulator does not allow you to define a load this way and then choose to ignore it. Most RAW files that PowerWorld has seen seem to be solved ignoring the loads that are in different areas than their terminal bus by using the "lines only" option. To overcome this, when PowerWorld reads a RAW file, [MW Transactions](#) are automatically created with the ID "RAW\_LOAD". (You can optionally choose to load the RAW file "with options". This will allow you to choose whether or not the transactions should be added. See [Case Formats](#) for more information.) These transactions are between the load's area and its terminal bus and are created so that the export from each area correctly matches a case solved using "lines only". If you know that your case was solved using "lines and loads", then click the button **Remove transaction due to tie-line loads**. Clicking this will delete all the MW Transactions with an ID of "RAW\_LOAD". You may also click the button **Add transactions due to tie-line loads** to recreate a list of transaction which represent flows between loads with different areas than their terminal bus.

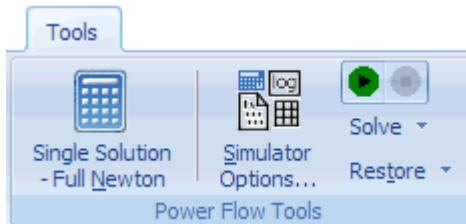
---

**Power Flow Tools Ribbon Group**


---

The main function of the PowerWorld Simulator is to simulate the operation of an interconnected power system. The simulation is accomplished by selecting **Single Solution – Full Newton** from either the [Quick Access Toolbar](#) or the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab. This activity performs a single Power Flow Solution. See [Solving the Power Flow](#) for more information.

The following tasks are also available from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab.


**Single Solution – Full Newton**

The main function of the PowerWorld Simulator is to simulate the operation of an interconnected power system. The simulation is accomplished by selecting **Single Solution – Full Newton** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab. This activity performs a single Power Flow Solution. See [Solving the Power Flow](#) for more information. Also note that if presently in Edit Mode, you will be automatically taken to Run Mode when choosing this.

**Simulator Options**

When choosing any of the solution methods, the options as specified in the [Simulator Options](#) will affect what calculation is performed. solves the power flow using the either the Full Newton AC load flow or the DC Approximation load flow, as specified in the [Simulator Options](#).

**Play Button**

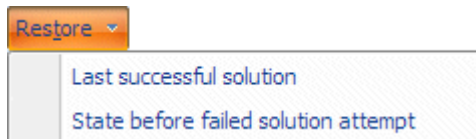
Click this button to cause repeated solutions of the system and to cause the animation of oneline diagrams

**Pause/Stop Button**

Click this button to stop the action of the Play button.

**Restore Menu**

The Restore menu offers option to either Restoring a Previous Solution or a Restore a Previous State



Sometimes a power flow attempt won't converge to a solution. When this occurs, the voltages and angles calculated by the solution engine will not satisfy the real and reactive power balance constraints at each bus. Then, the state currently stored in memory will not be an actual system operating point. It is often very difficult to coax the system to solve once it has failed to converge.

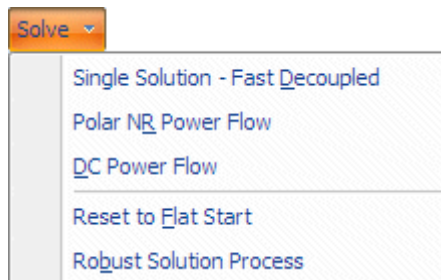
To help you recover from a solution attempt that has failed to converge (both timed simulations and single solutions), Simulator offers you two options.

After Simulator solves a system successfully, it will store the voltages and angles it found in memory. If the changes that you then make to the system result in a system that can't be solved, you can select **Restore > Last Successful Solution** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab to reload the results of the last converged solution. After reloading this information, Simulator will re-solve the system and refresh all displays.

In addition to restoring the last converged solution, Simulator also gives you the ability to restore the state of the system as it was just prior to the unsuccessful solution attempt. This can be thought of as "un-doing" the effect of the solution attempt. Before attempting a solution, Simulator stores the state of the system in memory. If it solves the power flow successful, Simulator will discard this pre-solution state. However, if the power flow fails to converge, Simulator will keep the state in memory. To recover it, select **Restore > State Before Solution Attempt** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab. Simulator will replace the non-converged post-solution state with the pre-solution state and refresh the displays. You can then play with the system to try to make it easier to solve.

If you are working with large systems, you should be aware that saving these system states can consume a lot of memory. Therefore, Simulator offers you the option to disable one of both of these features. To do this, select **Simulator Options** from the **Case Options** ribbon group on the [Options](#) ribbon tab to open the [Simulator Options Dialog](#), and then go to the Storage sub-tab of the Solution tab. You will see two checkboxes there that can be modified to control whether or not these extra system states are saved.

### Solve Menu



### Single Solution – Full Newton

The main function of the PowerWorld Simulator is to simulate the operation of an interconnected power system. The simulation is accomplished by selecting **Single Solution – Full Newton** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab, or from the [Quick Access Toolbar](#). This activity performs a single Power Flow Solution. See [Solving the Power Flow](#) for more information. Also note that if presently in Edit Mode, you will be automatically taken to Run Mode when choosing this.

### Polar NR Power Flow

This perform a Newton Raphson power flow solution using polar coordinates instead of rectangular coordinates as is normally done in Simulator.

### DC Power Flow

This automatically change the Simulator Options to use the [DC Approximation](#) and then immediately performs the DC power flow solution.

### **Reset to Flat Start**

Select **Solve > Reset to Flat Start** from the **Power Flow Tools** ribbon group on the [Tools](#) ribbon tab to initialize the Power Flow Solution to a "flat start." A flat start sets all the voltage magnitudes and generator setpoint voltages to 1.0 per unit and all the voltage angles to zero. Usually, a flat start should be used only if the power flow is having problems converging. You can also use the flat start option on the [Simulator Options Dialog](#) to initialize every solution from a flat start.

### **Robust Solution Process**

The Robust Solution Process provides a method to attempt to reach a solution when the standard load flow (Newton-Raphson) solution fails. The robust process performs a solution in a series of steps.

First, the robust solution will turn off all controls in the case. Then the load flow will be solved using a fast decoupled power flow. If the fast decoupled power flow reaches a solution, Simulator then immediately solves the load flow using the Newton-Raphson load flow, still keeping the controls turned off. If the Newton-Raphson solution is also successful, Simulator will begin adding controls back into the solution process, one type of control at a time. Thus the generator MVAR controls are added back in, and the load flow is resolved. Then the switched shunt controls are restored, and the load flow is again resolved. Simulator will continue in this manner by reintroducing next the LTC control, followed by the area interchange control, and lastly the phase shifter control. Furthermore, when reintroducing the phase shifter control, the controls are added one at a time for each phase shifter, with a load flow solution occurring after each.



---

## Solving the Power Flow

---

At its heart, Simulator is a Power Flow Solution engine. Power flow is a traditional power engineering calculation that is performed to determine the flows on all lines and the voltages at all buses in the system given the power injections at all buses and the voltage magnitudes at some of them.

The power flow problem entails solving a system of nonlinear equations. Solving a nonlinear system requires the use of an iterative algorithm to hone in on the correct solution. Many nonlinear system solvers have been developed, and PowerWorld provides access to the full Newton-Raphson method with an optimal multiplier and the fast decoupled method.

Usually, the power flow computation converges quickly. However, it is certainly possible to model conditions for which no Power Flow Solution exists, or for which the algorithm cannot converge to the solution within the [maximum number of iterations specified](#). For such situations, the [message log](#) will provide a message indicating that the computation failed to converge. Furthermore, unless [blackouts are disabled](#), the screen is grayed, and a message indicating a blackout has occurred is shown.

In order to calculate the power flow solution, simply press the Single Solution button on the main Program Toolbar. Click Single Solution actually results in three nested loops which solve the power flow: MW Control (Outer) Loop, Controller (Middle) Loop, and Power Flow (Inner) Loop.

### Power Flow (Inner) Loop

The Power Flow Loop is where the traditional power flow matrix equations are solved.

### Controller (Middle) Loop

Once the Power Flow Loop is solved, control devices look to see if their control requirements are being met. Control devices include load-tap-changing (LTC) transformers, phase-shifting transformers, switched shunts, DC transmission lines, and generators hitting max/min var limits. If any control devices requiring changes, then these changes are made and the power flow loop is resolved. This continues until no more control loop changes are made.

### MW Control (Outer) Loop

After the Control Loop has completed, the MW Control loop is entered and generation (and possibly load) is moved to meet the MW control options set in the case. Normally, MW control is done by [area control](#) with each area varying generation to meet its own load, losses and interchange. However, you may also use [island-based control](#) to dispatch MWs by island.

If any MW control is needed, then the Control Loop and Power Flow Loop interaction must be repeated and so on.

---

## Area Control

---

One of the most important aspects of interconnected power system operation is the requirement that each operating area changes its total generation to match changes in the sum of its load plus losses plus power transactions with other areas. This requirement is normally met by Automatic Generation Control (AGC). The purpose of AGC is to ensure that the actual MW output of an area is equal to the scheduled MW output of the area. The AGC system accomplishes this by first calculating the Area Control Error (ACE), which is defined as

$$ACE = P_{\text{actual}} - P_{\text{scheduled}}$$

In Simulator,  $P_{\text{scheduled}}$  for an area is made up of the Area's [MW Transactions](#) and the Area's *Unspecified MW Export*. MW Transactions represent the transfer of power between two areas in the power system. This transaction is done presumably under a contract between the two areas. The advantage of using MW transactions to describe these is that it ensures that the total export of all areas is consistent: if one area is exporting 100 then another area is automatically importing that power. However, each area also specifies a value called the Unspecified MW Export which can be entered on the [Area Information Dialog](#). The unspecified MW export represents an export of power from the area that goes to an unspecified other area. When using unspecified MW exports, it is important that the total unspecified MW exports in the system sum to zero. PowerWorld highly recommends that care be taken when using unspecified exports.

Whenever the ACE is greater than zero, it means that the area is over generating and thus needs either to decrease generation or to sell more. Likewise, whenever the ACE is less than zero, the area is under generating and thus needs either to increase generation or to buy more. AGC works to keep the ACE close to zero.

In Simulator, there are six options for implementing AGC:

### No area control

The output of the generators does not change automatically. You must manually change the generation to match system load/losses/transaction variation. All change in load/losses/transaction in this area will be made up at the islands slack bus.

### Participation Factor Control

The output of all the area's generators who have their AGC field set to "YES" change automatically to drive the area control error (ACE) to zero. Each generator's output is changed in proportion to its participation factor. Checking this option enables the **Set Factors** button on the [Area MW Control Options](#) tab of the [Area Information Dialog](#), which, when pressed, opens the [Generator Participation Factors Dialog](#). Participation Factor Control only adjusts generation when a change to the system has taken place, such as changing the amount of load in the case, or defining new area to area transactions.

In Participation Factor Control, the ACE is allocated to each AGC generator in the area in proportion to that generator's participation factor divided by the total of the participation factors for all AGC generators in the area. A generator's participation factor cannot be negative. By default, a generator's participation factor equals its current MW setpoint value, but individual participation factors can be changed.

### Economic Dispatch Control

The output of all AGC generators in the area changes automatically to drive the area control error (ACE) to zero. Each generator's output is changed so that the system is dispatched economically, based on cost information entered for the generators in the case. Note that cost data is not generally included in standard load flow data. Without realistic cost data

entered into Simulator, the use of the economic dispatch algorithm may not be very useful. Cost data must be obtained from another source and entered into a case in Simulator, either manually or through the use of [Simulator Auxiliary Files](#).

With Economic Dispatch (ED) Control, Simulator tries to change the output of the area's AGC generators economically so that the area's operating cost is minimized. ED control recognizes that some generators are less expensive than others and tries to use the least expensive generators to the largest extent possible.

To do economic dispatch, we need to know how much it would cost to generate one more MW at a particular generator. This is known as the incremental or marginal cost. For example, for the cubic cost curve model, the incremental cost for each generator is modeled using the formula:

$$\lambda_i = IC_i(P_{gi}) = (b_i + 2c_i P_{gi} + 3d_i (P_{gi})^2) * \text{fuelcost} \quad \$/\text{MWH}$$

The plot of  $IC_i(P_{gi})$  as a function of  $P_{gi}$  is known as the incremental-cost curve. The economic dispatch for a system occurs when the incremental costs for all the generators ( $\lambda_i$ ) are equal. This value is known as the system  $\lambda$  (lambda) or system incremental cost. Its value tells you how much it would cost to generate one more MW for one hour. The system lambda becomes important when trying to determine whether or not an area should buy or sell power. For example, if an area can buy power for cheaper than it can generate it, it might be a good idea for the area to buy power.

### Area Slack Bus Control

Only the output of the area's slack bus changes automatically to drive the area control error (ACE) to zero. This type of generation control is usually only good for small disturbances to the injections and/or transactions in a case, and can often fail to find a solution when larger disturbances are examined.

### Injection Group Area Slack Control

Only the Injection Group specified as the Injection Group Area Slack will change to automatically drive the area control error (ACE) to zero. This allows you to be very specific with each area as to which generation (or load) should vary to maintain ACE.

### Optimal Power Flow (OPF)

The [OPF](#) option will only be available if you have the OPF add-on for PowerWorld Simulator. The OPF control is very similar to the Economic Dispatch control in that it attempts to dispatch generation to minimize costs. The additional function of the OPF is to minimize the costs while also obeying line, transformer, and interface limit constraints. This option is also not useful without realistic generator cost information, which usually must be obtained from another source and entered into Simulator to augment a load flow case.

The OPF control also relies on the cost curve in order to perform an economically optimal power flow. However, the OPF routine makes use of piecewise linear curves in its solution algorithm. This does not prevent you from entering the cost information as cubic cost models, described by the equation above. Rather Simulator's OPF routine allows you to specify how to break up the cubic curve and model it as a piecewise linear curve for the OPF algorithm.

In addition, you can also enter piecewise linear curves directly instead of the cubic cost curve models. In fact, a mixture of piecewise linear and cubic models is acceptable. For the economic dispatch routine, whichever type of model is entered will be used directly for each generator. For the OPF routine, all piecewise linear curves entered directly will be used as is, and any cubic models entered will be converted to piecewise linear curves internally during the processing of the OPF algorithm.



---

## Set Generator Participation Factors

---

Participation factor control is another of Simulator's mechanisms for distributing an area's responsibility to serve its load, losses, and interchange. It is particularly well-suited to implementing [automatic generation control \(AGC\)](#) when you do not have good economic information for an area's generators. With participation factor control, the amount of power that each generator contributes to meeting its areas load, loss, and interchange responsibilities is controlled by the size of its participation factor. The unit that has the largest participation factor contributes the most, and the unit that has the smallest participation factor contributes the least.

The **Set Generator Participation Factors Dialog** gives you a convenient way to define the participation factors for multiple generators. You can set the participation factor according to a number of different formulae and then apply this prescription to all generators in a specific area, all generators in a specific zone, all generators in the system, or all generators whose display filters are currently set to true.

To display the **Set Generator Participation Factors Dialog**, you first need to open the [Area Information Dialog](#) and switch to the Area MW Control Options page. The Area Information Dialog has a button labeled **Set Participation Factors** that is enabled only if the Participation Factor Control option is selected under the Area Control Options heading. Set the area on participation factor control by selecting the Participation Factors option, and then press the **Set Participation Factors** option.

The Set Generator Participation Factors Dialog is divided into two parts. The first part, which occupies the top half of the form, allows you to indicate how the participation factors should be calculated or set for each generator. Your options include:

**Max MW Rating of Generator** The participation factor for each generator is set to the generator's maximum MW capability.

**Difference Between Max and Current Output** The participation factor for each generator is set to the generator's reserve power, so that each generator participates in proportion to how much it has left to contribute.

**Constant Value of** The participation factor for each generator is set to the same hard-coded value.

**File** The participation factor for each generator is read from a file. The first line of the file should contain the keyword NUMBERS or NAMES indicating whether generators are identified by bus number or by bus name in the file. All subsequent lines should be comma-delimited and contain three fields: the number or name of the generator's bus, the generator's id, and the generator's participation factor.

If you choose any of the first three options, you then must tell Simulator to what generators you want to assign the participation factors. To assign the participation factors to all generators in a specific area, select the **All Generators in Area** option, and then choose the area from the adjacent dropdown box. If you want to assign the participation factors to all generators in a specific zone, select the **All Generators in Zone** option, and then choose the zone from the adjacent dropdown box. If you want to assign the participation factor to all generators in the system regardless of their area or zone affiliation, select the **All Generators in System** option. Finally, if you want to assign the participation factor to just those generators whose display filter criteria evaluates to true, choose the **All Generators With Valid Display Filters** option.

If you instead chose to read participation factors from a file, only those generators whose factors you read from the file will have their factors set by this action. However, unless each generator's associated area is set to control generator output using participation factor control, this information will be ignored. To make sure that each generator's area is set to participation factor control, check the **Set Corresponding Areas to Participation Factor Control** box. Then, each corresponding area will be set to participation factor control.

---

## Transaction Dialog

---

The Transaction Dialog can be used to modify or create Base Interchange Transactions between two areas. This dialog can be opened by right-clicking in the Base Interchange table of the [Area Information Dialog](#) and choosing Show Dialog to see an existing transaction definition, or Insert to add a new transaction.

The transaction dialog is divided into two pages of controls:

### Information

#### Transacting Area

This is the "from" area for the transaction. For an export from this area, the transaction value will be positive. For an import into the Exporting Area, the transaction value specified would be negative. Flow out (export) of the exporting area is always considered positive.

#### Transaction To Area

This is the "to" area for the transaction.

#### Transaction ID

New in Simulator version 10 is the ability to have multiple transactions defined between the same two areas. Because of this, it is now required that transactions also have a Transaction ID.

#### Rename Transaction ID

If you wish to change the transaction ID for a particular transaction, enter the new value in the **Transaction ID** field, and press this button.

#### Switch Directions

Press this button if you wish to reverse the defined Exporter and Importer for the transaction.

#### Transaction MW Amount

The MW amount of the transaction being defined. This value should be positive for an export from the Exporting area to the Importing area. The value can also be entered as negative to define a transaction into the Exporting area from the Importing area.

#### Transaction Minimum MW

The minimum transaction amount between the two areas. This field is only enabled if the check box labeled **Transaction Dispatchable in OPF** is checked.

#### Transaction Maximum MW

The maximum transaction amount between the two areas. This field is only enabled if the check box labeled **Transaction Dispatchable in OPF** is checked.

#### Exports/Imports Transmission Charge

The cost to transfer power, in \$/MWh. This adds an economic penalty for making the transfer, making the transfer less likely to take place. Half the charge is assigned to the buyer and half to the seller.

#### Transaction Enabled

Transaction can now be defined and either enabled or disabled. Any disabled transactions will be ignored in both a standard power flow solution and an OPF solution.

#### Transaction Dispatchable in OPF

Checking this box enables the transfer to be dispatched by the OPF algorithm. Dispatching the transaction makes the two areas of the transaction appear to be one area for the purpose of economically dispatching the generation in the two areas. The transaction can have a maximum and minimum transfer amount when dispatchable, and a transmission charge associated with the transaction.

#### **Determine Price in OPF**

Checking this box allows the OPF algorithm to determine the cost associated with the transfer. The cost is determined by the marginal cost of enforcing the power balance constraint for the combined areas. This is the typical way to implement a transfer if *both* areas are on OPF control. If only one of the two areas are on OPF control, then the area which is off of OPF control needs to specify a price for the transfer. This is done by explicitly defining a piecewise linear cost curve.

#### **Piecewise Linear Transaction Cost Curve**

These two curves are only enabled if the option **Transaction is Dispatchable in OPF** is checked and the option **Determine Price in OPF** is unchecked. These two curves can be defined for the purpose of assigning a price to the transfer of power between one area on OPF control and another area which is not on OPF control. Separate curves can be defined for export transactions (from the Exporter to the Importer) and import transactions. To add points to the curves, simply right-click in the grid and choose Insert from the local menu. Enter the MW value and corresponding marginal cost for the inserted breakpoint of the piecewise linear curve you are defining. To delete a point, right-click on that row in the grid and choose Delete from the local menu.

#### **Custom**

The Custom page of the Transaction dialog is simply a location to log information about the transaction in the Memo box, and to see the values (if any) stored in some of the custom fields for the transaction. To log information about the transaction, simply switch to the Custom page on the dialog, and start typing your information or comments about the transaction in the page, or enter custom values in one of the custom fields.



---

## Calculate MW-Distance

---

Simulator can estimate MW \* Distance quantities for the system's areas and zones that result from a specified transaction. Given a transaction from a specified source to a specified sink, Simulator uses [power transfer distribution factors \(PTDFs\)](#) to estimate the change in flow for each line in the system that results from the transaction. For each line, multiplying the line's change in flow by its length then gives the MW \* Distance index for that line. Simulator then sums the MW \* Distance indices by area and by zone to obtain the total MW \* Distance for each area and zone in response to the specified transaction.

Because the MW \* Distance calculations use PTDFs, you must access the MW \* Distance functionality from the PTDF Dialog. Once you have calculated PTDFs for a particular transaction by pressing the **Calculate PTDFs** button, click the **Calculate MW \* Distance** button to bring up the **MW \* Distance Calculations Dialog**.

The top portion of the MW \* Distance Calculations Dialog is used to set the lengths of the lines in the case. Although line length is represented as a data element in the power flow case, it often is left blank. However, Simulator needs line length information if it is to calculate MW \* Distance indices. Simulator offers a few options regarding the source of line length information. If you do not have access to line lengths, either from the existing case or an external text file, Simulator can estimate line lengths for you. It does this by using the Ohms/Length values you specify in the table for lines of various kV. Simply indicate the voltage levels in the first row of the table, and the corresponding ohms or reactance per length in the second column. You do not need to differentiate here between English and metric units, because the calculation is independent of the measurement system. If you want the length estimates calculated using this table to overwrite any line lengths that may already be present in the case, be sure to check the **Always Estimate Length** checkbox; otherwise, the new estimates will set the lengths only of lines whose pre-defined length isn't greater than zero. If you want the estimates to populate the lengths of lines in the model so that, when you save the model, the estimated lengths are saved as part of the model, check the **Save Estimates With Case**. (This provides a handy way to set line lengths for a case that might not have any defined.) Note that, in performing these estimates, transformers are defined as having zero length. If you do not want Simulator to estimate line lengths but instead want to use the line lengths that are currently stored in memory, check the **Do Not Use Length Estimates** box. Finally, if you want to load line lengths from a text file, click the **Load Line Lengths from File** button. This file can be either comma- or space-delimited, and each line must have the following fields in the order specified:

From\_Bus\_Number To\_Bus\_Number Circuit\_ID Length

Once Simulator knows how to calculate line lengths, it can calculate MW\*distance indices for each area and zone. Specify the amount of MW that will be transacted in the **Size of Transaction** textbox. You may use the arrows to increase or decrease the size of the transaction. Simulator assumes that the transaction is to occur between the source and sink groups for which you just calculated PTDFs. Press the button labeled **Calculate** to compute the indices. Two tables are populated with the results of the calculation, one for areas, and another for zones. Use the tabs to switch between the two tables. These tables are [Case Information Displays](#) and thus share common to all case information displays. Thus, you can sort the tables, add or delete columns, access the area and zone dialogs, print the tables, and save their content as HTML.

Several options can be set to customize the calculation of MW\*Distance. These options are reached from the [MW\\*Distance Options Dialog](#).

---

## MW-Distance Options

---

The calculation of MW\*Distance quantities can be customized in a number of ways. These options are set from the MW\*Distance Options Dialog.

### Include Tie Lines Only

If this box is checked, then the only branches that contribute to the MW\*Distance calculation are those that tie two areas together. Otherwise, both tie lines and lines internal to areas and zones are included in the calculation. In the latter case, tie lines are assumed to belong to the area that owns the metered end of the branch.

### Internal Flows

If you choose to include both tie line flows and flows internal to areas and zones in calculating MW\*Distance quantities, you have two options for how to treat internal flows. You can ignore flows resulting from the transaction that flow in the reverse direction of the existing flow on a branch by checking the **Include flow increases only** checkbox. You can also choose to treat all such counterflows as negative contributions to an area or zone's MW \* Distance value by checking the **Deduct flow reductions** checkbox.

### Omit Branches

To omit branches for which the PTDF corresponding to the transaction is less than a specified value, specify a nonzero percentage in this textbox.

---

## Area Control Error (ACE) Chart

---

The ACE chart plots the area control error for an area over time. For details on ACE, please see topic [Area Control](#). To access this display, go to the [Tools](#) ribbon tab, and choose **Other > Charts > ACE Chart** from the [Other Tools](#) ribbon group. This option is only available in Run Mode. You may also press the corresponding button on the Simulation Summary tab of the [Area Information Dialog](#). If you use the ribbon to view this chart, it shows information for the first area in the case. The strip chart starts to plot the data when you open it, with new data appearing on the left. You can change the scale of either the x-axis (the time axis) or the y-axis (Scheduled Transactions axis) by right-clicking anywhere on the axis itself and specifying the new axis limits and number of intervals.

Right-click on the display (except on the axes) to view the display's local menu. The local menu is used to print the strip chart, save the strip chart in a file, copy the strip chart to the Window's clipboard, or change the number of the area being displayed.

Use the display's control-menu box to close the display (i.e. the 'X' button at the top right corner of the form).

---

## Area Load and Generation Chart

---

The Load and Generation chart plots an area's load + losses and generation in MW when you are animating (playing) and solving the load flow every animation cycle. To access this display, go to the [Tools](#) ribbon tab, and choose **Other > Charts > Area Load and Generation Chart** from the [Other Tools](#) ribbon group. This option is only available in Run Mode. You may also press the corresponding button on the Simulation Summary tab of the [Area Information Dialog](#). If you use the ribbon to view this chart, it shows information for the first area in the case. The strip chart starts to chart the data (if animation is running) when you open it, with new data appearing on the left. You can change the scale of either the x-axis (the time axis) or the y-axis (Load/Generation MW axis) by right-clicking anywhere on the axis itself and specifying the axis limits and number of intervals.

Right-click on the display (except on the axes) to view the display's local menu. The local menu is used to print the strip chart, save the strip chart in a file, copy the strip chart to the Window's clipboard, or change the number of the area being displayed.

Use the display's control-menu box to close the display (i.e. the 'X' button at the top right corner of the form).

---

## Area Losses Chart

---

The Area Losses chart plots an area's real power losses when you are animating (playing) and solving the load flow every animation cycle. To access this display, go to the [Tools](#) ribbon tab, and choose **Other > Charts > Area Losses Chart** from the [Other Tools](#) ribbon group. This option is only available in Run Mode. You may also press the corresponding button on the Simulation Summary tab of the [Area Information Dialog](#). If you use the ribbon to view this chart, it shows information for the first area in the case. The strip chart starts to chart the data (if animation is running) when you open it, with new data appearing on the left. You can change the scale of either the x-axis (the time axis) or the y-axis (Load/Generation MW axis) by right-clicking anywhere on the axis itself.

Right-click on the display (except on the axes) to view the display's local menu. The local menu is used to print the strip chart, save the strip chart in a file, copy the strip chart to the Window's clipboard, or change the number of the area being displayed.

Use the display's control-menu box to close the display (i.e. the 'X' button at the top right corner of the form).

---

## Area MW Transactions Chart

---

The Scheduled MW Transactions chart plots the scheduled real power (MW) transactions for an area when you are animating (playing) and solving the load flow every animation cycle. To access this display, go to the [Tools](#) ribbon tab, and choose **Other > Charts > Area MW Transactions Chart** from the [Other Tools](#) ribbon group. This option is only available in Run Mode. You may also press the corresponding button on the Simulation Summary tab of the [Area Information Dialog](#). If you use the ribbon to view this chart, it shows information for the first area in the case. The strip chart starts to chart the data (if animation is running) when you open it, with new data appearing on the left. You can change the scale of either the x-axis (the time axis) or the y-axis (Scheduled Transactions axis) by right-clicking anywhere on the axis itself and specifying the new axis limits and number of intervals.

Right-click on the display (except on the axes) to view the display's local menu. The local menu is used to print the strip chart, save the strip chart in a file, copy the strip chart to the Window's clipboard, or change the number of the area being displayed.

Use the display's control-menu box to close the display (i.e. the 'X' button at the top right corner of the form).

---

## Area Average Cost Chart

---

The Average Cost per MWH chart plots the average cost per MWH for an area over time. This value is calculated by dividing the total cost of operating the area (generation cost + purchased power cost - revenue from power sales) by the MW load in the area. To access this display, go to the [Tools](#) ribbon tab, and choose **Other > Charts > Area Average Cost Chart** from the [Other Tools](#) ribbon group. This option is only available in Run Mode. You may also press the corresponding button on the Simulation Summary tab of the [Area Information Dialog](#). If you use the ribbon to view this chart, it shows information for the first area in the case. The strip chart starts to plot the data when you open it, with new data appearing on the left. You can change the scale of either the x-axis (the time axis) or the y-axis (Scheduled Transactions axis) by right-clicking anywhere on the axis itself and specifying the new axis limits and number of intervals. Note that as the area's load increases, the average cost per MWH tends to increase.

Right-click on the display (except on the axes) to view the display's local menu. The local menu is used to print the strip chart, save the strip chart in a file, copy the strip chart to the Window's clipboard, or change the number of the area being displayed.

Use the display's control-menu box to close the display (i.e. the 'X' button at the top right corner of the form).

---

## Bus View Display

---

The **Bus View Display** feature provides a graphical display which allows you to quickly browse information about a bus and everything connect to that bus. The bus view displays enable convenient bus-by-bus navigation through the power system. From the bus view, you can find out a bus' voltage and angle, the load, shunt compensation, and generation connected to the bus, and the flows on all lines emanating from the bus. You can also discover the bus' area and zone affiliations, as well as the bus' marginal cost. Moreover you can find out all information about the elements associated with the bus by directly invoking their associated information dialogs. The advantage of the bus view displays, is that they are auto-created oneline diagrams.

Along the top of the bus view display resides a panel of controls. The buttons labeled **Back** and **Ahead** allow you to step through the history of buses you have viewed thus far. To move the bus view to a particular bus, type a number or name into the edit box next to the caption **Bus**. If a number is typed the case will be scanned for a bus of that number, otherwise the case will be scanned for a bus that starts with the string entered. If you're unsure which bus that you are looking for click the **Search For...** button to bring up a display that allows you to search using wildcard strings for the bus. There are also drop-downs in this top panel for **Options >** and **Views >**. These are discussed below.

Below this top panel sits the actual bus display. The bus you have chosen to inspect, which we shall refer to as the *target bus*, is represented by a long, thick black horizontal line. Notice that the bus' voltage in kV and per unit, its angle, and its marginal cost are specified to the left of the bus. Any loads and generating loads connected to the target bus are drawn above the bus symbol, along with their associated annotation. Emanating from the bottom of the bus symbol are all transmission lines and transformers that connect the target bus to its neighbors. The transmission line and transformer symbols are equipped with pie charts and annotation identifying flows as measured at the target bus, as well as arrows to identify the direction of MW flow on the branch. Neighboring buses are represented as filled rectangular regions, with symbol indicators included if other types of devices, such as loads, generators, etc., are attached. When you drag the mouse over one of the neighboring bus symbols, it turns into a pointing finger. Clicking the left mouse button when the mouse cursor is in this shape redefines the target bus to be the bus whose symbol you just clicked. The bus view display is redrawn to show the same sort of display for the newly chosen target. You can go back to the previously displayed target bus by clicking the **Back** arrow, and then return to this record by clicking the **Ahead** arrow.

It is useful to think of the bus view displays as nothing more than an additional oneline diagram. In other words, you interact with the objects drawn on the bus view display in the same way you work with objects on a more conventional Simulator oneline. Right-clicking on any power system object will bring up that object's local menu, which includes a link to the object's associated information dialog. As on a conventional oneline diagram, flows on a bus view display can be animated. Right-clicking on the bus view display's background will generate the same local menu as other oneline diagrams.

The bus view display can be generated using any of the following methods:

- Go to the [Case Information](#) ribbon tab, and click on the **Bus View** from the **Views** ribbon group
- Go to the [Onelines](#) ribbon tab, and click on the **Bus View** from the **Views** ribbon group.
- Right click on the bus of interest on the oneline diagram to display the bus' local menu, and choose Bus View. The bus view display will open with the selected bus already displayed.



- From any of the case information displays that convey bus information, right click on a record to bring up its local menu, and choose Bus View Online. The bus view display will open with the corresponding bus already displayed.
- Click the **Bus View** Button on the [Case Information Toolbar](#).

To switch between the bus view and the main online, use the **Open Windows Menu** available on the **Views** ribbon group on the [Case Information](#) ribbon tab, [Case Information](#) ribbon tab, or [Windows](#) ribbon tab. To close the bus view display, simply close the form using the **X** button in the top right corner of the bus view display.

## Bus View Options

### Number of Tiers

The bus view can display one or two "tiers" of buses in the display. Use this selection from the **Options** menu to toggle the number of tiers displayed.

### Show Hints

When this option is checked, holding the cursor over an object will briefly pop up a hint box containing information about that object.

### Show Serial Buses

When putting in the branch connections, the Bus View display will look out into the network and find the next bus which has more than two neighbors. It will then make this the destination bus for that branch of the Bus View. The intermediate buses will then be shown in order above the destination branch. This option works especially well in systems with a lot of multi-section lines.

### Show Equivalent Lines

This option indicates to the Bus View display whether or not to include branches representing equivalent circuits as connections in the display.

### Represent Multi-Section Line Objects

This option indicates to the Bus View display whether or not to draw multi-section line display objects when appropriate. If this option is not checked, then the intermediate buses of the multi-section lines will be rendered normally. If this option is checked then the intermediate buses of multi-section lines will not be rendered on the bus view and only the terminals of the multi-section line will appear.

### Default Drawing Values

Choosing this option will open the Default Drawing Values for New Objects dialog. Changing these options can change some of the drawing aspects of the bus view, including device color and font size or color.

### Open Multiple Bus Views

This option indicates whether to open multiple bus views simultaneously. Choices are never, always, and prompt for confirmation when a new additional bus view is about to open.

### Include Field Labels

Selecting this option will place labels for each displayed field on the bus view diagram.

### Change Bus Link Color

Selecting this option will display the color palette to select the color with which the buses linked to the current bus will be displayed.

### Show Field Suffixes

This option specifies whether to display the field values units as suffixes. If this option is not selected, all the fields will be display as a value without units.

### **Dynamic Formatting**

Click on this to bring up the [Dynamic Formatting](#) for case information displays, and all views and onelines.

## **Views**

### **Define Custom View**

The fields displayed on the bus view can be customized using this option. Clicking on this option will open a customization settings display, in which you can add and remove field definitions for the objects on the bus view display. Customized bus view layouts can be saved with the case for recall, identifiable by a custom bus view layout name. Custom layouts can also be saved to a file for loading into another load flow case.

### **Input Data**

Switching the bus view to Input Data changes the bus view from displaying system state information to displaying input data information. For example, switching to Input Data view will display line impedances and limits, generator minimum and maximum outputs, etc. The default Input Data view fields can be modified using the Define Custom View customization dialog.

### **System State**

Switching the bus view to System State changes the bus view from displaying Input Data information to displaying system state information. This will result in line flows being displayed, voltage and angles displayed, etc., of the current solution state of the system. The default System State view fields can be modified using the Define Custom View customization dialog.

---

## Substation View Display

---

The Substation View Display feature is analogous to the [Bus View Display](#).

Along the top of the substation view display resides a panel of controls. The buttons labeled **Back** and **Ahead** allow you to step through the history of substations you have viewed thus far. Substation View options are available from the **Options** menu. To find a substation using search tools, click on the **Search For** button. The next two controls following the Substation label allow to specify a substation name (in the first text box) or a number (in the second text box). If you type a number or name that does not exist, the substation display will continue displaying the current substation.

Below this top panel sits the actual substation view display.

Just as with the bus view, it is useful to think of the substation view displays as nothing more than a oneline diagram. In other words, you interact with the objects drawn on the substation view display in the same way you work with objects on a more conventional Simulator oneline. Right-clicking on any power system object will bring up that objects local menu, which includes a link to the object's associated [information dialog](#). As on a conventional oneline diagram, flows on a substation view display can be animated. Right-clicking on the substation view display's background will generate the same [local menu](#) as other oneline diagrams. The substation view can be generated using any of the following methods:

- Go to the [Case Information](#) ribbon tab, and click on the **Substation View** from the Views Ribbon Group
- Go to the [Onelines](#) ribbon tab, and click on the **Substation View** from the **Views** ribbon group.
- Right click on the substation of interest on the oneline diagram to display the substation's local menu, and choose Substation View. The substation view display will open with the selected substation already displayed.
- From any of the case information displays that convey substation information, right click on a record to bring up its local menu, and choose Substation View Oneline from the Substation Records menu. The substation view display will open with the corresponding substation already displayed.
- Click the **Substation View** Button under the **Records** menu on the [Case Information Toolbar](#).

To switch between the substation view and the main oneline, use the **Window** menu tree on the main menu. To close the substation view display, simply close the form using the **X** button in the top right corner of the substation view display.

---

## **Chapter 11: Run Mode Tools and Options**

---

Run mode provides a number of commands for simulating and modifying the case that are different from Edit Mode.

This chapter covers the following:

- General Tools
- Contingency Analysis
- Fault Analysis
- Contouring
- Distribution Factors
- Sensitivities

---

## Generator Economic Curves

---

Four characteristic curves describe the efficiency and resulting costs associated with operating a particular generating unit. These four curves plot

- Fuel Cost
- Heat Rate
- Input-Output
- Incremental Cost

Simulator can display plots of all these curves. To display a particular plot for a generator, right-click on the generator in Run Mode to display its local menu, and then select the plot you wish to see. The plot will be presented in its own window. The windows for all plots exhibit identical characteristics. For example, the current operating point is identified by a red filled circle. Right clicking on an open area of the window displays the plot's local menu which allows you to print the plot, save it to a file, or copy it to the clipboard for use in other programs. To adjust the length and number of intervals shown on an axis, right-click on the *axis* (not on the *numbers*) then specify the min and max display values and number of intervals. To close a plot window, simply click the X button in its top right corner.

The Run Mode generator local menu also provides access to a fifth type of plot curve - the "All Area Gen IC Curves" plot. This plot simply shows the incremental cost curves and present operating points of all generators in the same area as the generator on which you clicked.

### Fuel Cost Curve

The fuel cost curve specifies the cost of fuel used per hour by the generating unit as a function of the unit's MW output. This is a monotonically increasing convex function.

### Heat-rate Curve

The heat rate curve plots the heat energy required per MWH of generated electrical output for the generator as a function of the generator's MW output. Thus, the heat rate curve indicates the efficiency of the unit over its operating range. Generally, units are least efficient at the minimum and maximum portions of their MW output capability and most efficient somewhere in the middle of their operating range. The vertical axis is plotted in MBtu/MWH and the horizontal axis is plotted in MW. You may interpret the heat rate for a generator producing X MW as follows: the heat rate indicates the amount of heat input energy per MWH of generation required to produce X MW of power. The lower this number, the less input energy is required to produce each MWH of electricity.

### Input-Output Curve

The input-output curve is derived simply from the heat-rate curve by multiplying it by the MW output of the unit. This yields a curve showing the amount of heat input energy required per hour as a function of the generator's output.

### Incremental Cost Curve

By multiplying the input-output curve by the cost of the fuel in \$/MBTU, one obtains the cost curve for the unit in \$/hr. By taking the derivative of the cost curve, one obtains the incremental cost curve, which indicates the marginal cost of the unit: the cost of producing one more MW of power at that unit.

---

## Find Branches that Create Islands

---

To access the Find Branches that Create Islands, go to the [Tools](#) ribbon tab, and choose **Connections > Find Branches that Create Islands** from the [Other Tools](#) ribbon group.

There are several options for selecting which ac lines to process.

### Line Processing Options

#### All ac Lines

All ac lines in the power system model will be processed.

#### Use Area/Zone/Owner Filter

All ac lines that meet the defined Area/Zone/Owner Filters will be processed.

Select Area/Zone... to display the Area/Zone/Owner Filters dialog.

#### Use Selected

All ac lines that have the Selected? field set to 'YES' will be processed.

Click Select Lines... to display all ac lines and change the Selected? field.

#### Meets Filter

All ac lines that meet a selected advanced filter will be processed.

Use the drop down box to select a defined [advanced filter](#) or click Define Filter to display the Advanced Filters for Branch Dialog. This dialog will allow you to define a new advanced filter for a branch or update an existing filter.

By checking **Do not display radial lines creating a single bus island** those lines that only island a single bus will not be displayed with the results.

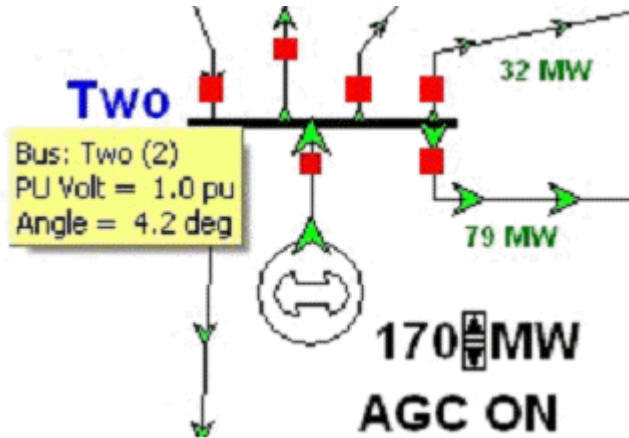
Click **Determine Branches** to start the processing once all options have been set.

Because the processing of ac lines in a large power system may take some time, there is an **Abort** button that is enabled once the line processing has started that will stop the processing at any point.

The list of resulting ac lines that create islands will be displayed under **Branches that Create Islands**. To show the list of buses that are islanded by an outage of any line in the list, select a line and the **Islanded Buses** list will be populated. Both the Branches that Create Islands list and the Islanded Buses list are [case information displays](#) and have the same [local menu options](#) and characteristics of other case information displays.

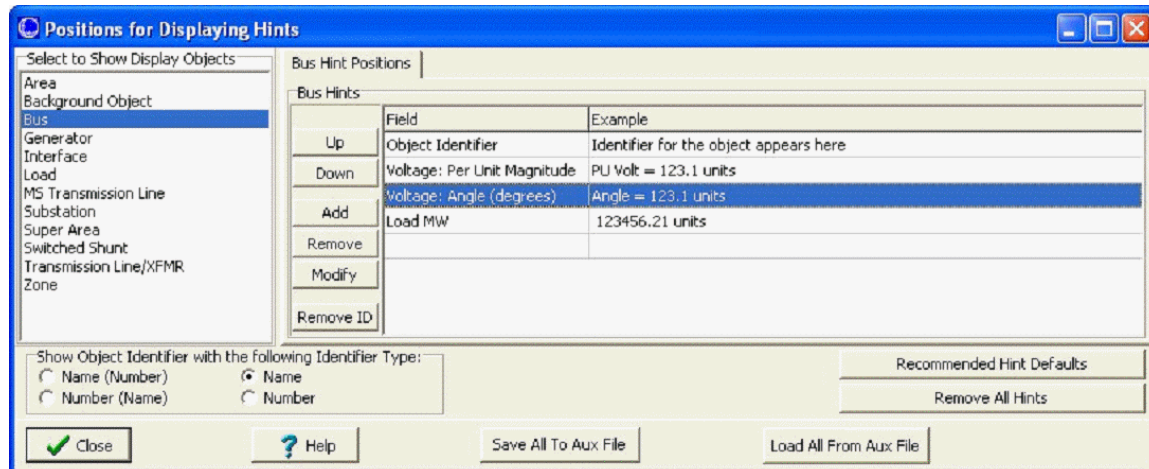
## Custom Hint Values

Simulator can be configured to show a short hint string when the mouse is hovered over objects on the oneline diagrams. This hint string will appear in a yellow box as shown below.



This option must be set first by opening the [Simulator Options dialog](#) and then going to the **Online** category and finally checking the **Show Online Hints**. The user may then customize what text and values appear in the hint by choosing **Custom Hint Values** on the **General Options** ribbon group of the [Options](#) ribbon tab. These **General Options** are also available on the [Online](#) ribbon tab.

This brings up a dialog on which customizations are made for each type of display object that can show a hint.



Choose the type of object on the left side of the dialog. With this display object chosen, choose which fields of this dialog you would like to include as part of the custom hint. Click the **Add**, **Remove**, or **Modify** buttons to add, remove, or modify a field to the hint. Click the **Up** and **Down** buttons to change the order in which the fields appear. On the dialog that appears after clicking Add or Modify, you may customize the field with a prefix, modify the digits and decimal points, and specify whether to show the units as a suffix. The Example column shows an example of what text will appear for the hint.

The *Object Identifier* is also available that will appear as the first line of the hint if chosen. To remove this identifier, click the **Remove ID** button. When the object identifier is not being shown the **Remove ID** button caption will change to **Add ID**. Click the **Add ID** button to add

the object identifier back to the hint. You may specify the format of the object identifier by setting the **Show Object Identifier with the following Identifier Type** option appropriately.

The hint customizations are stored with the PowerWorld Binary file (PWB) and in the Windows Registry. The customizations affect all oneline diagrams (including the Bus View and Substation View). The customizations may also be stored in an [Auxiliary File](#) for use in moving the customizations between computers and cases. Click the **Save All to Aux File** button to save all these customizations to an Auxiliary File.



---

## Line Loading Replicator

---

The Line Loading Replicator will attempt to set the loading on a transmission line or interface to a desired real power flow amount. This is accomplished by calculating distribution factors for a selected group of loads and generators to determine how the real power injection of these elements can be adjusted to produce the desired flow. If the desired flow cannot be met, the new flow amount that can be achieved and the injection changes required to meet this flow will be reported. Minimum and maximum MW limits for loads and generators will be enforced when calculating the injection changes.

To open the Line Loading Replicator, go to the [Tools](#) ribbon tab, and choose **Line Loading Replicator** from the [Run Mode](#) ribbon group. This is only available in [Run Mode](#).

### Select Device

#### Device Type

Select whether to adjust the flow on a transmission line/transformer or an interface. All devices must already be defined in the case.

#### Device Identifier

Specify the near bus, far bus, and circuit identifier for a branch or the interface name and number for an interface. Branches will be monitored in the near bus to far bus direction and interfaces will be monitored in the direction in which they are defined to be monitored.

#### Present Flow

Present real power flow on the selected device.

#### Desired Flow

Desired real power flow on the selected device. The difference between the Desired Flow and the Present Flow (Desired Flow – Present Flow) will be the flow change for which injection changes will be calculated.

### Available Injection Groups

#### Injection Group Identifier

Specify the injection group name and number that contains the loads and generators whose injection can be changed to reach the Desired Flow. The injection group must already be defined in the case. The injection group definition is used simply for selecting the group of elements to be used in the calculations; the participation factors defined with the participation points in the injection group are not used in the Line Loading Replicator tool.

#### Only Include AGCable Generation and Load

If checked, only those loads and generators in the selected injection group whose AGC status is *YES* will be included in the calculations.

### Calculation Method

Select the method used to calculate the distribution factors used to determine real power injection changes.

### Max and Min Load Limits for Injection Changes

#### Use Max and Min Load Values

If checked, the maximum and minimum load values will come from the maximum and minimum values defined with each individual load record.

#### Set Max and Min Load Values

Click this button to show the [Load Records Case Information Display](#). The maximum and minimum load values can be defined by displaying the **Max MW** and **Min MW** fields and setting the values as appropriate.

#### **Use Multiplier on Present Value**

If checked, the minimum and maximum value for each load will be determined by applying the **Min Multiplier** and **Max Multiplier** to the present load value.

#### **Calculate Injection Changes**

Click this button to calculate the injection changes required to meet the Desired Flow on the selected device.

#### **Injection Changes**

This is a case information display that will provide a summary of the injection changes needed, and the elements that can effect these changes, in order to meet the Flow Achieved on the selected device. The Flow Achieved will differ from the Desired Flow if there are not enough injection changes available to meet the Desired Flow.

The results are reported in terms of injection. For generators, a positive injection means an increase in generator output and a negative injection means a decrease in generator output. For loads, the opposite is true; a positive injection means a decrease in load and a negative injection means an increase in load.

**Injection Change**, **Present Injection**, and **New Injection** fields will be shown by default. These Present Injection and New Injection fields are both calculated based on the present injection of the given element with the New Injection value also being dependent on the Injection Change value. The Present Injection and New Injection values will be updated any time that the present injection for an element changes in the power system. This is important to keep in mind if making any changes to the power flow case while the Line Loading Replicator dialog is open.

Injection changes are calculated so that generator and load limits are enforced. Which load limits are used is determined in the **Max and Min Load Limits for Injection Changes** option settings. The generator limits used are the minimum and maximum limits defined with each generator. These limits can be accessed from the [Generator Case Information Display](#). The load and generator limits can also be displayed in the Injection Changes table by adding the **Min MW** and **Max MW** fields to the table.

#### **Total Injection Increase/Decrease**

This value is the amount of injection increase and decrease required to meet the Flow Achieved. The net injection will be zero.

#### **Flow Achieved**

Flow amount that can be achieved by implementing the Injection Changes listed. This value will differ from the Desired Flow if there are not enough injection changes available to meet the Desired Flow.

#### **Implement Injection Changes**

Click this button to make changes to the power flow case to match the calculated Injection Changes. Automatic generation control (AGC) and phase shifter control are disabled for the entire case when choosing to implement changes.

#### **Implement Injection Changes and Solve Power Flow**

Click this button to make changes to the power flow case to match the calculated Injection Changes and then solve the power flow. The power flow is solved with the settings that are specified in the [Power Flow Solution Options](#) specified with [Simulator Options](#). When

choosing to implement changes, these options are updated so that automatic generation control (AGC) and phase shifter control are disabled globally for the case. The Present Flow value is updated after the completion of the power flow solution.

---

## Dynamic Formatting Overview

---

The [Dynamic Formatting dialog](#) will allow the specification of how a graphical object will be rendered depending on the power system object that is representing. The graphical object refers to objects in the oneline diagram, in bus and substation views, as well as to some parameters of the case information displays.

The case and the one-line diagrams will have each a list of Dynamic Formatting settings. The Dynamic Formatting settings defined in the case are always applied to bus and substation views, and optionally they can be applied to the case information displays. The one line diagrams will use its own settings, but optionally can use the general settings defined with the case.

In general, the oneline dynamic formatting settings have a higher priority, followed by the case dynamic formatting settings, and this can't be reversed. Also, inside each of the list of dynamic formatting settings, a priority can be specified, so that objects can be rendered according to the settings with the highest priority.


In the case of the oneline diagrams, the dynamic formatting settings will only be applied during Run Mode. However, these settings can be modified at any time, without regard for the mode.

---


## Dynamic Formatting Dialog

---

The Dynamic Formatting dialog for the active oneline can be accessed by:

- Selecting **Dynamic Formatting** >  **Active Oneline** on the **General Options** ribbon group of the [Options](#) ribbon tab. These General Options are also available on the [Oneline](#) ribbon tab.
- Selecting **Dynamic Formatting (Active Oneline)** from the [oneline local menu](#), or

The Dynamic Formatting dialog for the general case information displays, bus and substation views, and for all onelines can be accessed by:

- Selecting **Dynamic Formatting** >  **Case Info / All Views And Onelines** on the **General Options** ribbon group of the [Options](#) ribbon tab. These General Options are also available on the [Oneline](#) ribbon tab.
- Selecting **Dynamic Formatting (All Views)** from the [bus view local menu](#) or the [substation view local menu](#), or

This dialog presents the following options:

### Set Format Priority

It is possible to have multiple dynamic format definitions created and stored with a case and/or active oneline diagram. This option allows you to set a format priority order for all of the dynamic format definitions, so that any diagram objects that meet more than one dynamic filter will use the settings of the dynamic filter with the highest set priority.

### Allow Oneline to use dynamic formatting defined with case

This option will only be available when the dynamic formatting settings correspond to an active oneline. It indicates whether or not the oneline diagram will use the dynamic formatting definitions specified in the case.

### Object Type

The type of power system object with graphical representation, such as a bus, a load, a generator, etc.

### Filter Criteria

The filter that applies to the object type defined. The Dynamic Formatting settings will apply to the graphical object only if the corresponding power system object meets the specified filter. If the criteria box is empty, the Dynamic Formatting settings will assume that all the objects of the specified type meet the criteria.

### Formatting Active

If this option is unchecked, the dynamic formatting definition will be ignored when the objects are rendered. Otherwise, it will be applied if there are graphical objects whose characteristics match the rest of the characteristics defined in this dialog.

### Force visibility

When this option is checked, the objects will be displayed (assuming the dynamic formatting applies to them) independently of the visibility of the layer to which such objects belong, and with no regards of the low and high zoom levels.

### Show Lookup Tables

Checking this box will display the **Use Lookup** options with the characteristic settings. The **Use Lookup** option with certain field characteristics allow you to define a table of different field values and corresponding characteristic values. If you check Show Lookup tables, and then check the Use Lookup field next to one of the characteristics, you will open the Lookup

definition table. This table allows you to select a field for the Object Type you have selected. This field is the field that you define values for in the left column of the lookup table. To insert entries in the lookup table, right-click in the table grid and choose Insert from the popup menu. This will place a default record in the table. Modify the Field Value that Simulator should look for in the table, and a corresponding Characteristic Value that should be used for the field value given. For example, you could use a lookup table with Line Thickness for transmission line objects, where the thickness of the line would vary based on the amount of flow on the line. Different thicknesses could be assigned to different levels of flow using a lookup table.

### **Context**

In the Context Objects, the user will specify what specific type of graphical objects the dynamic formatting will apply to. This list view will be populated with the Case Information Display object (if the dynamic formatting definitions set correspond to the general case), plus the several graphical objects related to the Object Type. (For buses, for example, it will include the graphical bus, the bus fields, and the bus gauge).

### **Fields**

In the Fields list view, the user will be able to select which fields the dynamic formatting settings will apply to. This view will be populated only for those context objects with fields, such as the Case Information Display and the Object Fields. This list is only visible when you choose certain types of context objects in the context objects list.

### **Show Only Commonly Used Fields**

If this option is checked, only a reduced list of selected fields will be displayed. Otherwise, all the fields belonging to the object will be shown.

### **Characteristics**

The characteristics that the user will be allowed to modify dynamically. These include line thickness, style, color, and background color; font name, size, color, and style; highlight color; surround shape, color and thickness; color and magnification of the 'X' on top of the objects; and blinking color and interval.

---

## Difference Flows

---

The **Difference Flows** feature provides an easy mechanism for comparing two power system cases. For example, Difference Flows can be used to show the difference in transmission line flows and bus voltages resulting from a contingency or a change in power transfer between two areas.

The Difference Flows Dialog can be accessed by:

- Go to the [Tools](#) ribbon tab, and choose **Difference Flows** > **Difference Flows** from the [Run Mode](#) ribbon group. This is only available in [Run Mode](#).
- Selecting **Difference Flows** from the [online local menu](#) in Run Mode.

Use of this feature affects all aspects of the Simulator environment. When using the Difference Flows tool, information shown on oneline diagrams, case information displays, and [power flow lists](#) is governed by which of the three [Difference Flow Case Types](#) (base, current or difference) is currently being displayed.

All Difference Flows Actions are controlled via the Difference Flows Dialog or the drop-down list on the Difference Flows toolbar button.

---

## Difference Flows: Case Types

---

When using the Difference Flows tool, information shown on oneline diagrams, case information displays, and [power flow lists](#) is governed by which of the three case types is currently being displayed.

The case types available are:

- **Base Case** – A solved power system that serves as the reference for the difference flows tool. To establish a base case, set up a solved power system corresponding to the desired operating point. Open the Difference Flows Dialog and click the button labeled **Set Present as Base Case**.
- **Present Case** – The operating point used in the Difference Flows comparison. **Note:** The Present Case must have the same numbering scheme as the Base Case for proper operation of the Difference Flows tool. See [Topological Differences](#) for more information.
- **Difference Case** – The difference between the Present Case and the Base Case values. The values displayed in the Difference Case are established using the Base Case as the reference.

To toggle between the different case views, open the [Difference Flows Dialog](#) and select the desired case type. Alternatively, you can click the **Difference Flows** button in the Run Mode ribbon group on the [Tools](#) ribbon tab and select the desired case type.



---

**Difference Flows Dialog**

---

To display the Difference Flows dialog:

- Go to the [Tools](#) ribbon tab, and choose **Difference Flows** > **Difference Flows** from the [Run Mode](#) ribbon group. This is only available in [Run Mode](#).
- Select **Difference Flows** from the [online local menu](#) in Run Mode.



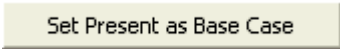
---

## Using Difference Flows

---

To use the Difference Flows tool:

- Set up a solved power system corresponding to a desired operating point. This operating point will be defined as the [Base Case](#).
- Select **Difference Flows** from the Run Mode ribbon group on the [Tools](#) ribbon tab, or select **Difference Flows** from the [oneline local menu](#) in Run Mode.
- On the **Difference Flows Dialog**, click the button labeled **Set Present as Base Case**. This stores the current operating point as the Base Case.



- Define the operating point ([Present Case](#)) for which to perform the difference flows comparison. The Present Case may be developed either by modifying the Base Case as desired and re-solving, or by opening a new case selecting **Open Case** from the [Application Button](#). In the latter situation, the new case that you open must have the same bus numbering scheme as the Base Case.
- See [Difference Flow Case Types](#) for information on toggling between case views. The currently displayed case type is shown in the [PowerWorld Simulator Status Bar](#). When viewing either the Base Case or Difference Case, the corresponding status bar display will be highlighted.

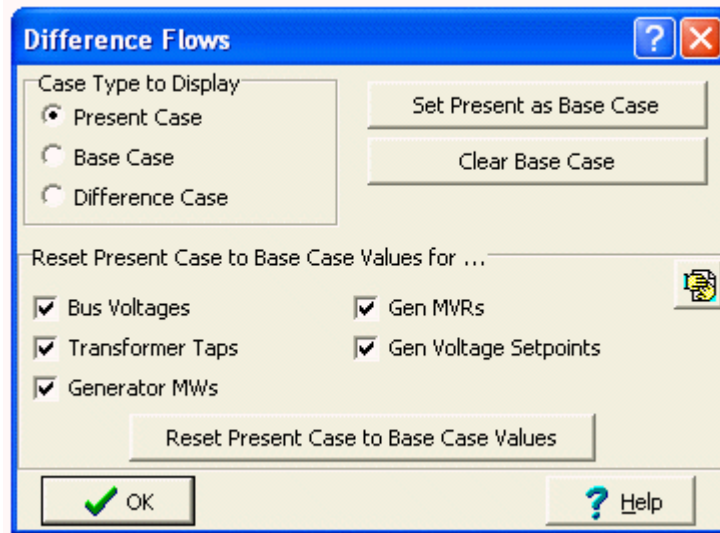


Note that the Difference Flows tool can only be used in Run Mode and the status bar will not display a case type while in Edit Mode.

When showing the Difference Case, most of the fields shown on the onelines and case information displays show the difference between the present value and its Base Case value. For example, on the Generator Records Display, an entry of 0.0 in the MW field indicates that the real power output of the generator did not change. An entry of 10.0 in the MW field indicates that the present real power output of the generator is 10 MW greater than it was in the Base Case.

At any time during a simulation, you can set the present case as the Base Case by clicking the corresponding button on the Difference Flows Dialog.

Conversely, if you have made changes to the present case, and you wish to revert to some or all of the base case values, you can click on the **Reset Case...** button. When you click this button, the dialog will expand to show you options for resetting specific types of values to their base case values.



**Expanded Difference Flows Dialog**

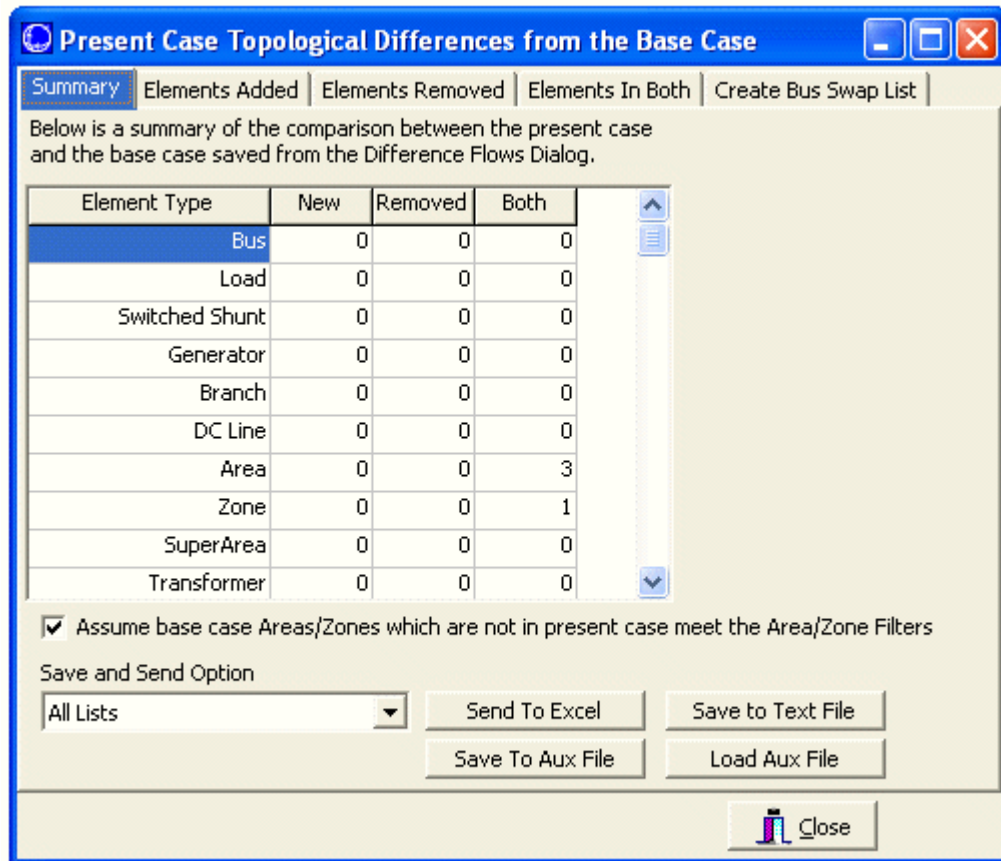
Once you have indicated which types of values you want to reset (by default, all are selected), you can click on the **Reset Present Case to Base Case Values** button to complete the process of reverting to base case values. Note: once you have finished resetting to the base case values, you can hide the Reset options by clicking the Hide Reset Button in the upper right hand corner of the Reset options panel.

The oneline diagrams and case information displays cannot indicate structural differences in the case very well, such as the addition or removal of a device. To identify such differences, make use of the [Present Topological Differences from Base Case](#) tool to identify topology differences.

## Present Topological Differences from Base Case

The Present Topological Differences from Base Case option provides the users a way to compare the topological differences between two difference cases in Simulator.

To compare two cases topologically, you must load the first case you wish to use as the reference into Simulator. Use the [Difference Flows](#) tool to set the case as the base case in memory. Once the reference case has been stored as the base case, open the second or comparison case into Simulator. Now if you check the [Tools](#) ribbon tab, and choose the **Difference Flows Menu** on the [Run Mode](#) ribbon group, you will see the option **Present Topological Differences from Base Case**. This is only available in [Run Mode](#). Choosing this option will open the dialog to display the topological differences.



Topological Differences Dialog

The Topological Differences Dialog contains five tabs. All but the Summary tab are instances of Case Information Displays and thus exhibit characteristics and controls similar to other displays.

### Summary

A listing of the types of objects in the case. The first column displays the number of NEW devices in the comparison case that were not in the base case. Alternately, the Removed column displays the number of devices in the base case that do NOT exist in the comparison case. The third column simply displays the number of items that were matched between the two cases.

**Elements Added**

Tabular listings of all objects that exist in the comparison case, but not in the reference case.

**Elements Removed**

Tabular listings of all objects that exist in the reference case, but are not present in the comparison case.

**Elements in Both**

Tabular listings of all objects present in both cases.

**Create Bus Swap List**

Used for setting up a bus renumbering list. It is possible that discrepancies in topology between the comparison case and the reference case can be due to a difference in bus numbering between the two cases. [Renumbering the buses](#) in the comparison case may take care of most of the topology discrepancies that are reported in this instance.

---

## Governor Power Flow

---

The Governor Power Flow dialog shows all information related to solving a governor power flow. The dialog is accessed by selecting **Other > Governor Power Flow** from the [Other Tools](#) ribbon group on the [Tools](#) ribbon tab. Using this dialog, you can modify settings related to solving the case while on Governor or "Island-Based Automatic Generation Control (AGC)".

The Governor Power Flow Dialog has two tabs: [Options](#) and [Generator Options](#)

---

**Governor Power Flow: Generator Options (Ignore Area/Zone/Owner filter) Tab**

---

This tab sheet presents a case information display with all the generators of the case. The columns shown in the display are helpful when looking at solving a Governor Power Flow.

---

## Governor Power Flow: Options Tab

---

This section describes the options available on the Options tab of the Governor Power Flow dialog. The dialog is accessed by selecting **Other > Governor Power Flow** from the [Other Tools](#) ribbon group on the [Tools](#) ribbon tab.

This tab sheet presents the following controls:

### **Disable Automatic Generation Control**

Disables automatic MW generation changes during the power flow solution.

### **Island AGC Tolerance**

The ACE mismatch tolerance allowed for the Island to be considered solved in terms of generation dispatch.

### **Island-Based Automatic Generation Control**

Select how the generations should be controlled for the dispatch.

- Disabled: The area and super area dispatch settings from the case will be used
- Use Participation Factors of individual generators: Each generator will have its own participation factor, and will contribute according to its participation factor divided by the sum of all participation factors of all other generators in the same island.
- Calculate Participation Factors from Area Make Up Power Values: Each area is assigned a "factor" as to how it should participate towards the generation dispatch in the island. This is similar to participation factors for generators. The total percentage the area contributes towards the generation change needed in the island is equal to its individual factor divided by the sum of all area make up power factors. Then within each area, the generator participation factors determine how the area's percentage is made up of available generation within the area. The area make up power values can be set in the table of areas on the right hand side of the dialog.
- Dispatch using an Injection Group: The island dispatch will be made up by a combination of generators and loads, defined in an [injection group](#) that can be selected from the dropdown list.

### **How should reactive power load change as real power load is ramped?**

If you are using an injection group with load as part of the dispatch, then you can specify how the reactive power load should respond as the real power demand of loads changes with the dispatch. The reactive power can either be kept at the starting ratio of real and reactive power of the load, or the MVAR amount can change at each load by a specified power factor.



---

## Contingency Analysis: An Introduction

---

Contingency analysis is a vitally important part of any power system analysis effort. Industry planners and operators must analyze power systems covering scenarios such as the long-term effects on the transmission system of both new generation facilities and projected growth in load. Market analysts and planners must make informed decisions regarding transactions for energy trade - whether that trade is for the next hour or months down the road. PowerWorld Simulator's Contingency Analysis tools provide the ability not only to analyze a power system in its base case topology, but also to analyze the system that results from any statistically likely contingent scenario.

Industry planning and operating criteria often refer to the n-1 rule, which holds that a system must operate in a stable and secure manner following any single transmission or generation outage. In PowerWorld Simulator, the individual contingency conditions can also be tailored to consist of either a single element (such as the loss of a transmission line or transformer), or multiple elements (such as the loss of a generator, several buses and a number of branches simultaneously). See [Available Contingency Actions](#) for a complete list of possible contingency actions.

Simulator can be set to use a Full Newton solution or use a DC Load Flow method to analyze each contingency. The Full Newton approach is not as fast as a DC Load Flow, but the results tend to be significantly more accurate and allow for gauging voltage/var effects.

The [Tutorial](#) is a great place to start learning about using Simulator's Contingency Analysis Tool. We also recommend reviewing the [Terminology](#) used throughout the Contingency Analysis help files prior to continuing.

---

## Available Contingency Actions

---

The current edition of Simulator can process lists of contingencies including the following:

**For Transmission Lines and Transformers (Branches)**

- The opening or closing of transmission lines and transformers

**For individual Generators, Loads, or Switched Shunts**

- Loss or recovery of a particular generator, load, or switched shunt

**For all the Generation, Load, or Switched Shunt at a particular bus**

- Movement of generation, load, or switched shunt MWs or Mvars to another terminal bus
- Changing or setting of load, switched shunt, or generator MWs or Mvars
- Change or setting of a generator or switched shunt voltage setpoint

**For Buses**

- Opening of all lines connected to a bus

**For Interfaces**

- Opening or closing of all lines or transformers in an interface

**For Injection Groups**

- Opening, closing, or changing of output of all devices in an injection group
- Setting or Changing the net MW Injection by a Percent or MW value and by proportion or by merit order

**Series Capacitors**

- Bypassing or placing series capacitors in service
- Setting of the Series Reactance by percent or to a particular per unit value

**Phase-Shifting Transformers**

- Changing or setting the Phase-Shifter Regulation MW value

**DC Transmission Lines**

- Opening or closing DC Lines
- Changing DC Line setpoint MW or Amp values

**Three-Winding Transformers**

- Opening or closing all legs of three winding transformers

**Solving the Power Flow**

**Contingency Block**

---

**Contingency Analysis: Terms**

---

**Contingency** – Contingencies are the basis of the Contingency Analysis tool. A single contingency can contain a single contingency element (referred to as n-1 contingencies) or multiple contingency elements.

**Contingency Action** - A statistically likely condition that could occur during power system operation. See [Available Contingency Actions](#) for a listing of actions supported by Simulator.

**Model Criteria** – Criteria under which a contingency action will occur. For example, the user can specify that a generation outage only occur if the pre-contingency flow on a line is higher than a specified amount. Simulator allows you to define Model Criteria, which consist of both [Model Conditions](#) and [Model Filters](#). Normally, no Model Criteria will be specified for a given action.

**Contingency Element** – Consists of a single Contingency Action and its associated Model Criteria, Status and Comment (optional). Multiple Contingency Elements can be defined for a single Contingency.

**Contingency Definition** – A listing of the Contingency Elements assigned to a Contingency.

**Global Action** – A [Global Action](#) is a list of contingency elements that occur for ALL contingencies. The elements included in a Global Action do not have to be entered as individual elements in each contingency.

**Contingency Block** – A list of Contingency Actions. A [Contingency Block](#) can be applied to individual Contingencies by including the block as a Contingency Element.

---

## Contingency Solution Options Dialog

---

By default, the contingency analysis will use the same options as the power flow algorithm when solving each contingency. You may also override these options for all contingencies, and/or for a specific contingency. This results in the ability to set the power flow solution options in contingency analysis at three different levels

1. Contingency Specific Options (see [Contingency Definition Dialog](#))
2. Contingency Analysis Options (see [Contingency Options Tab](#))
3. General Power Flow Solution Options (see [Power Flow Solution Options](#))

When Simulator executes a particular contingency, it will first look at options specified for that contingency. Any options which are defined for this contingency will be used. Other options set as *Use Default* will look to the Contingency Analysis Options. Again, any options which are defined for contingency analysis will be used. Finally, options marked in the Contingency Analysis Options as *Use Default* will be set to the same setting as the power flow solution options.

In order to specify options for a specific contingency, you click on the **Define/Modify Solution Options** button on the [Contingency Definition Dialog](#). In order to specify options for all contingencies, you click on the **Define/Modify Solution Options** button on the [Contingency Options Tab](#). Both of these will bring up the Contingency Solution Options Dialog.

This dialog contains many options regarding the power flow solution. For options which are a numerical value, just specify a new value to use. For options which are specified by a check-box, the check-box will have three settings: *use option*, *do no use option*, and *use default*. For a more detailed explanation of each option see the [Power Flow Solution Options](#).

To set the values to be the same as the present power flow solution options, click the **Set same as for Power Flow** button. To set all options back to use default, click the **Clear All Settings** button.

---

## Contingency Case References

---

Contingency Analysis always stores a Reference State or pre-contingency state. The Reference State stores information pertaining to:

- Buses
- Switched Shunts
- Limit Groups
- Loads
- Branches
- Generators
- Areas / Super Areas
- MW Transactions
- Power Flow Solution Options

See [Reference State Information](#) for details on the specific information stored.

The reference state is loaded into memory prior to the execution of each contingency during automatic processing of the contingency list. This ensures that all contingencies start from a common Base Case. Furthermore, the system is set back to the reference state following completion of the automatic processing. Note: The system is not restored to the reference state when the **Solve Selected Contingency** option is selected from the Contingency Record Display's local menu (see [Reference State Solution Options](#) for more information).

The reference state is always stored in Simulator after the first instance of opening the contingency analysis form. Therefore, opening the contingency analysis again may result in a prompt from the program. This prompt will ask you if you wish to set the contingency analysis reference state to the current state of the system (in case you have made changes since the last contingency analysis run), or if you wish to keep the existing contingency analysis reference state (which was set by previously opening the contingency analysis dialog.) Note that if you choose the second option, any changes you may have made to the case outside of the contingency analysis will be lost, as the reference state stored with the contingency analysis tool will reset the system state to the reference state.

Click [here](#) for information on defining the reference state.

---

## Contingency Case References - State Information

---

Simulator stores the following information with the Contingency Analysis Reference State:

### Bus State

- In or out of service
- Voltage Magnitude and angle
- Boolean expression stating whether any load exists at the bus (this is used because some of the contingency actions such as *MOVE GEN* will create a fictitious load if there is no generation at the destination bus)
- MW Marginal Cost

### Switched Shunt State

- In or out of service
- Nom value Mvar
- Control mode (FIXED/DISCRETE/CONTINUOUS)
- Nom Value MW
- All Setpoint Values
- Description of Blocks
- Low/high range for voltage control

### Limit Group State

- Rating sets for normal operation (Line, Interface...A, B, etc...)

### Load State

- In or out of service
- Constant power MW and Mvar components of load
- Constant current MW and Mvar components assuming one per unit voltage
- Constant impedance MW and Mvar components assuming one per unit voltage
- MW Scale
- Mvar Scale
- AGC Status
- Min/Max Load MW

### Line State

- In or out of service
- Bypassed?
- Transformer Control?
- Tap ratio
- Phase shift
- High/low desired setpoints
- Series Reactance X (only for series capacitors because a contingency element can change this)

### DC Line State (& Multi-Terminal DC Line State)

#### Generator State

- In or out of service
- MW Output
- Mvar Output
- Max/Min MW Output
- Participation Factor
- Max/Min Mvar Output
- Voltage Setpoint

- AGC Status (YES/NO)
- AVR Status (YES/NO)
- Capability Curve
- Use Capability Curve?
- Line Drop Compensation Impedance
- Line Drop Compensation Status (YES/NO/POSTCTG)

**Area State and Super Area State**

- Unspecified MW Transactions
- MW Scale
- Mvar Scale
- AGC Status
- Use Area Participation Factors? (for Super Area)

**MW Transaction**

- MW Value
- Enabled Status

**Power Flow Solution Options**

---

**Contingency Case References - Defining the Reference State**

---

The reference state is initially defined as the power system state that exists when the contingency analysis is opened for the first time for a given power flow case during a Simulator session. See [Reference State Information](#) for details on the specific information stored in the reference state.

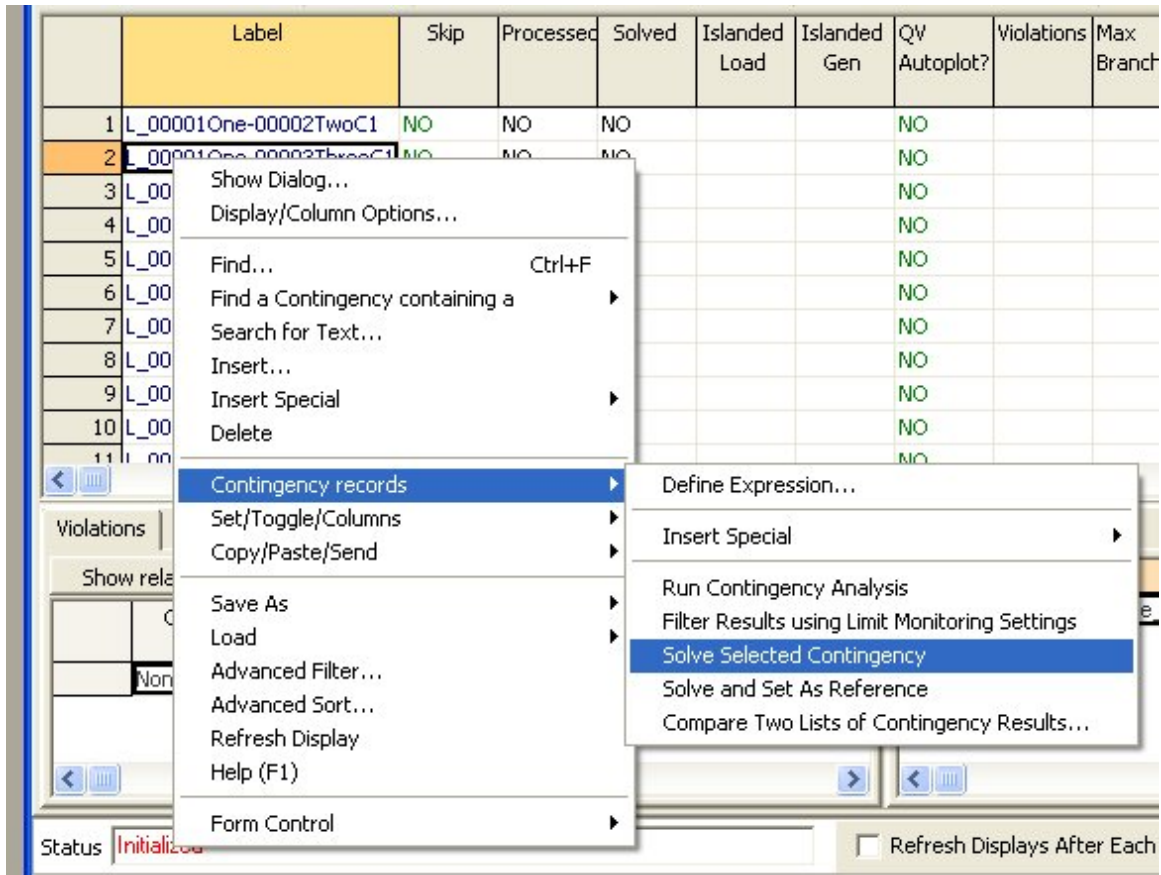
**Setting the reference state**

The reference state can be changed each time that the Contingency Analysis Dialog is opened after having already established a reference state. The [Setting Reference State When Contingency Analysis is Opened](#) option is available for determining how the reference state should be set when the dialog is re-opened during a Simulator session.

To change the reference while the Contingency Analysis Dialog is open, select the **Set as Reference** option from the [Other >](#) button on the [Contingencies Tab](#) of the Contingency Analysis Dialog. The reference state can also be redefined using the Solve and Set as Reference option from the Contingency Record Display's local menu (See [Reference State Solution Options](#) for more information).



## Contingency Case References - Reference State Solution Options



### Contingency Record Display Local Menu

When you solve contingencies one at a time, you may choose between the **Solve Selected Contingency** and **Solve and Set As Reference** options from the [Contingency Record Display's local menu](#).

**Solve Selected Contingency** causes Simulator to first load the reference state into memory then solve the contingency. Following the solution, *the reference state is not restored*; the system state then reflects the power system flows of the post-contingency state. The advantage of this approach is the ability to implement a contingency and then modify the system looking for possible actions that might mitigate violations caused by the contingency. Be aware; however, that prior to solving another contingency, Simulator will reset the system state to reference state thereby removing all modifications made following the previous contingency solution. The user may also automatically restore the system state to reference state by selecting **Other > Restore Reference** from the Contingency Analysis Dialog.

**Solve and Set As Reference** acts the same as Solve Selected Contingency with one exception. After executing the contingency, the post-contingency state is automatically set as the reference state. As a result, all subsequent contingencies will use the post-contingent state as the Reference State.

Click [here](#) for details on the specific information stored in the reference state.

Click [here](#) for information on defining the reference state.

---

### Contingency Analysis: Defining Contingencies

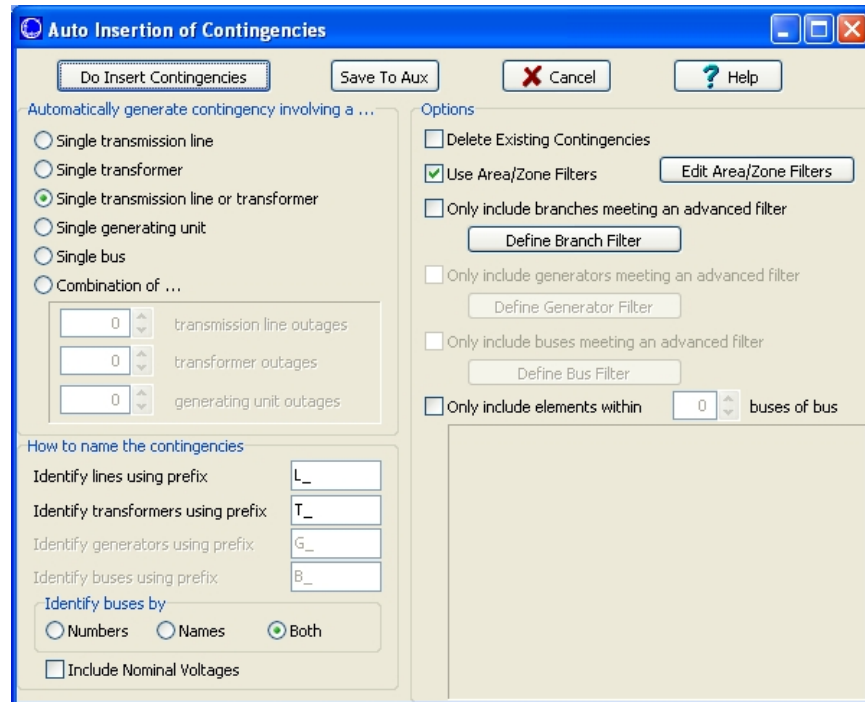
---

There are four options for defining contingencies. The user may: [Load Contingencies from a File](#), [Auto Insert Contingencies](#), or use the local menu to either **Insert** contingencies or **Quick Insert a Single Element Contingency**.

## Auto Insert Contingencies

Simulator allows you to automatically generate a contingency list containing branch, generator and/or bus outages. To accomplish this, click the **Auto Insert** button along the bottom of the [Contingency Analysis Dialog](#) or right-click on the list of contingencies table in the [Contingencies Tab](#) and select **Insert Special > Auto Insert Contingencies....** This opens the **Auto Insertion of Contingencies Dialog** (shown below).

When automatically inserting contingencies, you must specify the type, options and naming conventions you want for the new contingencies. You must also specify whether to delete or retain existing contingencies.



**Auto Insertion of Contingencies Dialog**

The Auto Insertion of Contingencies Dialog has the following controls:

### Do Insert Contingencies

Press the **Do Insert Contingencies** button to generate the contingency list once all other options have been set.

### Automatically generate contingency involving a...

The options available in this box define what to add to each automatically inserted contingency element. You can choose single transmission line, transformer, transmission line or transformer, generating unit, or bus contingencies. Choosing one of these options results in each contingency containing only one element of the specific type.

You can also define contingencies containing multiple outages by checking the **Combination of...** option, and then specifying how many of each type of element (Lines, Transformers, and Generating Units) you want considered in the contingency. When you use

the **Combination of...** option, Simulator will automatically determine all possible combinations for the element types specified (based on the settings in **Options**) and create the contingencies. Combination contingencies do not currently allow the inclusion of bus contingencies.

## Options

When Simulator auto-generates the contingency records for the element types specified, the options in this section will further determine which elements are included and which elements are ignored.

### Delete Existing Contingencies

When checked, any previously existing contingency records will be deleted before any automatically created contingencies are inserted.

### Use Area/Zone Filters

When checked, the elements included in the auto-generated contingencies will be only elements that are within areas and zones defined in the [Area/Zone/Owner Filters dialog](#). You may edit the Area/Zone/Owner filters by clicking on the **Edit Area/Zone Filters** button.

### Only include ... meeting an advanced filter

When checked, the branches, generators, or buses included in the auto-generated contingencies will be only those elements meeting conditions outlined by an [advanced filter](#). To set the conditions to be used, click the respective **Define... Filter** button.

### Only include elements within X buses of bus

When checked, only elements that are electrically within X number of buses from the specified bus will be included when creating the contingencies. For example, consider bus 1 is electrically connected to bus 2, which is in turn connected to bus 3. If we specify the bus to be bus 1, and choose to include only elements that are within 0 buses of bus 1, then the contingency record will include the branch between buses 1 and 2, and if desired any generators attached to bus 1. However, the branch between buses 2 and 3 and any devices attached to bus 2 and 3 will NOT be included in the contingency because bus 2 is electrically 1 bus away from bus 1.

To specify the bus used, you can find the bus by using the search engine. The search engine allows you to search by name or number. If you know the bus number, choose Sort by Number, and type the bus number in the search box. If you know the name of the bus, choose Sort by Name, and type the name of the bus in the search box. If you are not sure of the name of the bus, you can use wildcard characters to search through the list of buses until you find the desired bus.

## How to name the contingencies

This section allows you to define how each automatically inserted contingency record will be named.

### Identify ... using prefix

These four fields allow you to set a specific prefix for generators, lines, transformers and buses so that you can easily determine what type or types of contingencies are modeled in the auto-generated contingency records. By default, the prefixes are G for generators, L for lines, T for transformers and B for buses. However, you can change these prefixes to any character or set of characters you wish.

### Identify buses by

This field allows you to specify whether each contingency is labeled using the bus numbers, bus names, or both as identifiers. Whichever type of identifier you choose here

will be combined with the defined prefixes to uniquely define the individual contingency elements within each auto-generated contingency record.

**Include Nominal Voltages**

When this check box is checked, the nominal voltage of buses will be included in contingency labels.

---

## Loading Contingencies from a File

---

Simulator can load contingency definitions from a text file. The contingencies may be specified in one of three formats:

- [Simulator Auxiliary File Format \(\\*.aux\)](#) (also see [Contingency Subdata](#))
- Simulator Version 5-7 Contingency File Format (\*.ctg) (see the old users manual, or contact PowerWorld Corporation)
- [PTI PSS/E-formatted Contingency Files \(\\*.con\)](#).

To load contingencies from a text file, click the **Load** button along the bottom of the [contingency analysis dialog](#). A dialog box will be provided for you to specify the file from which to load the contingency records. Specify the file type in the **Files of Type** dropdown box, and select the appropriate file. If contingency records have already been defined for the case with which you are working, you will be asked if you wish to delete the existing contingencies. Respond affirmatively to delete the existing contingencies before adding the new ones from the specified files. Otherwise, click **No**, and the contingencies loaded from the file will be appended to the already existing list.

Simulator is also capable of loading bus throw over data from [PTI PSS/E-formatted load throw over files](#).

---

**PSS/E Contingency Format**

---

Simulator can read and write parts of the contingency format used by Power Technology's PSS/E. The current version of Simulator does support all parts of this format, except it does not recognize PTI's Automatic Contingency Specification flags. If you need Simulator to support these keywords, contact PowerWorld Corporation to express your need. Otherwise, we recommend you make use of Simulator's [Auto-Contingency creation tool](#).

---

### **PSS/E Load Throw Over Files**

---

Simulator can read [load throw over](#) settings used by Power Technology's PSS/E. These files typically have the extension of \*.thr or \*.dat. Note, however, that any modifications made to load throw over settings cannot be saved back to the PSS/E throw over file (only read access is currently available).



---

## Saving Contingency Records to a File

---

Simulator can save contingency definitions to a text file. To save contingencies:

- Click the **Save** button on the [contingency analysis dialog](#)
- Select **Save As...** from the [local menu](#) of the [contingency records display](#) (only useful for saving to Simulator's Auxiliary File Format).
- Click the **Save to Aux** button on the Auto Insertion of Contingencies Dialog.

The contingencies may be specified in one of three formats

- [Simulator Auxiliary File Format \(\\*.aux\)](#) (also see [Contingency Subdata](#))
- Simulator Version 5-7 Contingency File Format (\*.ctg) (see the old users manual, or contact PowerWorld Corporation)
- [PTI PSS/E-formatted Contingency Files \(\\*.con\)](#).

To specify the format for the contingency file, set the **Save As Type** option accordingly.

Note that there are limitations when saving to the Simulator Version 5-7 format or the PTI PSS/E formatted files.

### Limitations on the Simulator Version 5-7 format

- Does not support the actions SET or CHANGE.
- Does not support the action MOVE, except for Loads.
- Does not support any actions regarding an Interface
- Does not support any actions regarding a Bus

### Limitations on the PTI PSS/E format

- Does not support any actions regarding an Interface

---

## Global Actions

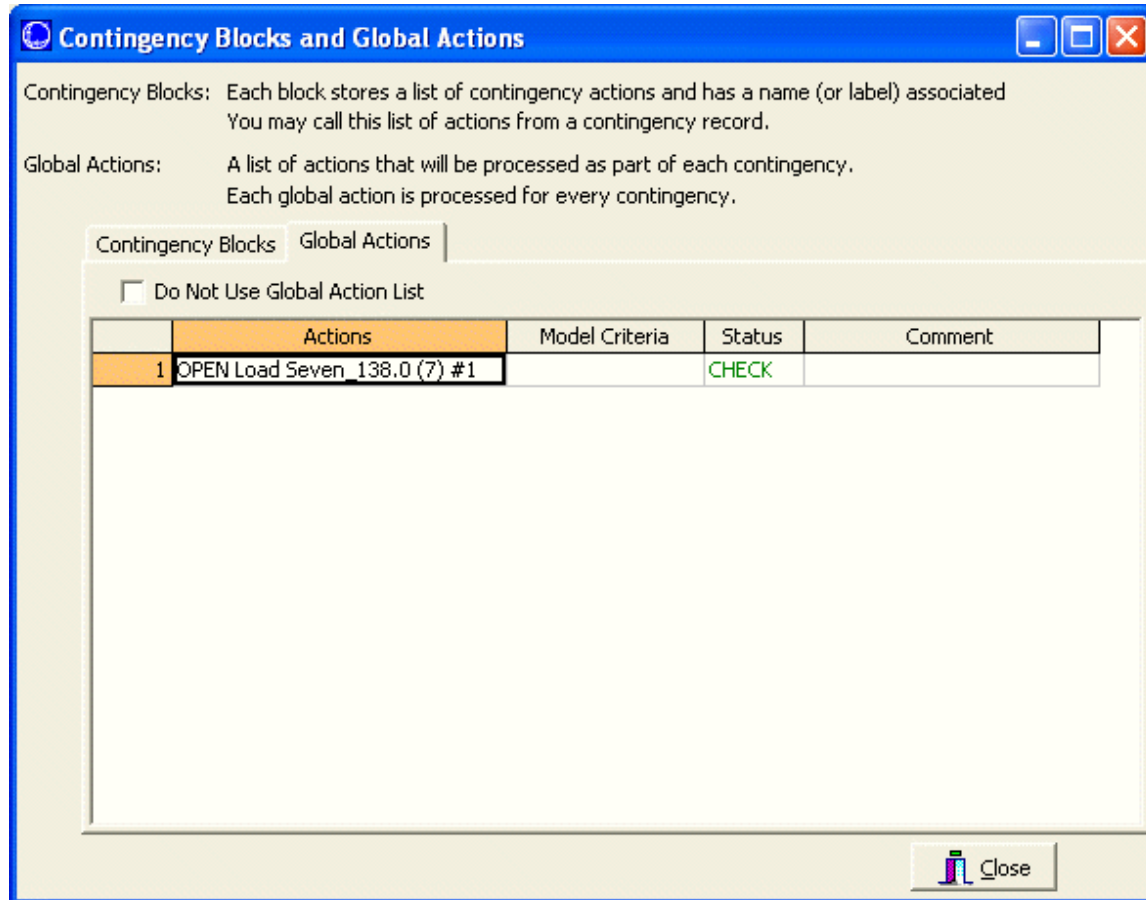
---

Global actions allow you to define a list of contingency elements that occur for ALL contingencies and do not have to be entered as individual elements in each contingency. These contingency elements are defined and processed in the same manner as contingency elements defined with specific contingencies.

Global actions are defined by clicking the **Contingency Blocks and Global Actions** button on the [Advanced Modeling](#) tab of the [Options](#) page of the Contingency Analysis dialog. Click on the **Global Actions** tab of the Contingency Blocks and Global Actions dialog to define Global Actions. Adding global actions occurs in a manner identical to that of the [Contingency Definition Dialog](#). Right-click in the Actions list to open the [Contingency Element Dialog](#), which allows you to add contingency elements to be treated as Global actions.

The format of the string that describes the actions and the **Model Criteria** and **Status** fields are set in the same manner as described in the [Contingency Definition Display](#).

To disable the use of the Global Actions, check the **Do Not Use Global Action List** option.



---

## Contingency Blocks

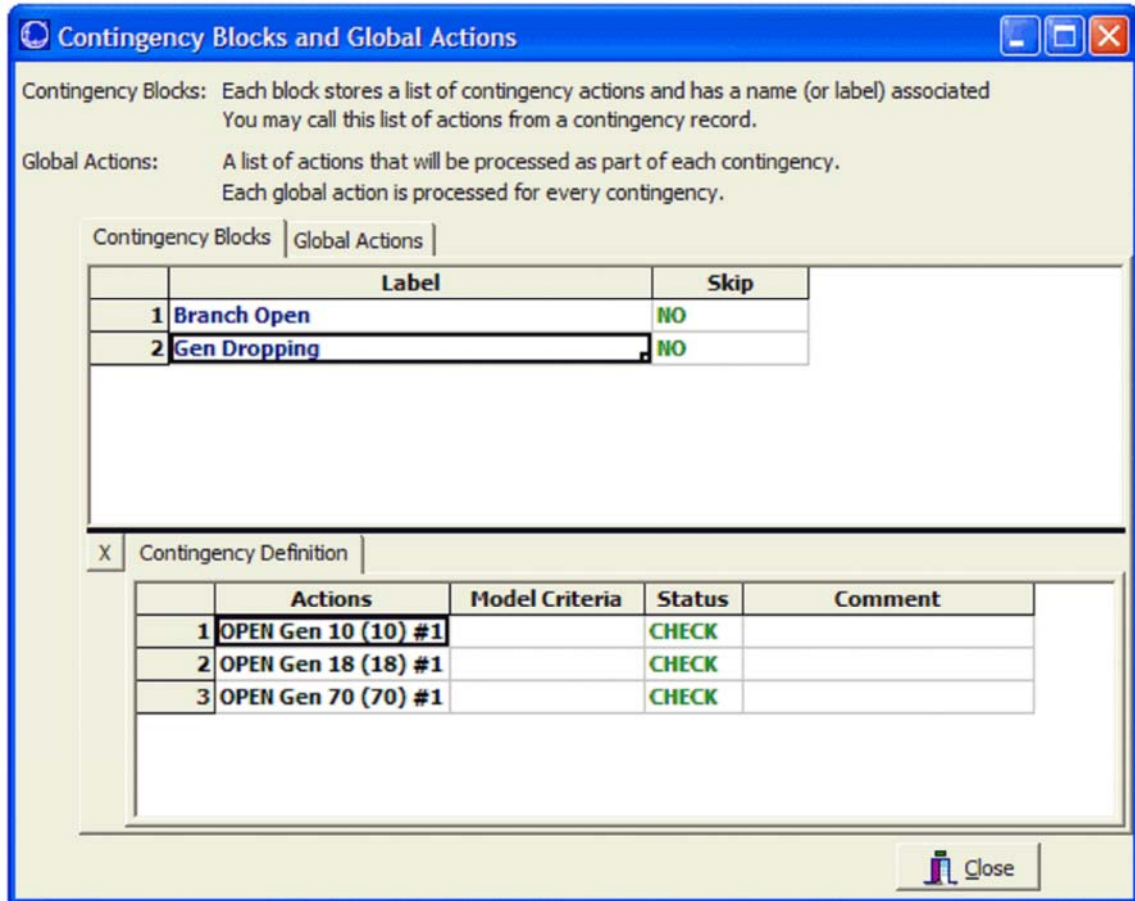
---

Contingency Blocks are very similar to a [Contingency Record](#), however no results can be associated with them. Contingency Blocks consist of a list of contingency actions. The block is then given a name so that any Contingency Record can call on a Contingency Block. When a contingency block is included as part of a contingency, the Contingency Record will incorporate all the actions from the contingency block into the actions performed by the contingency.

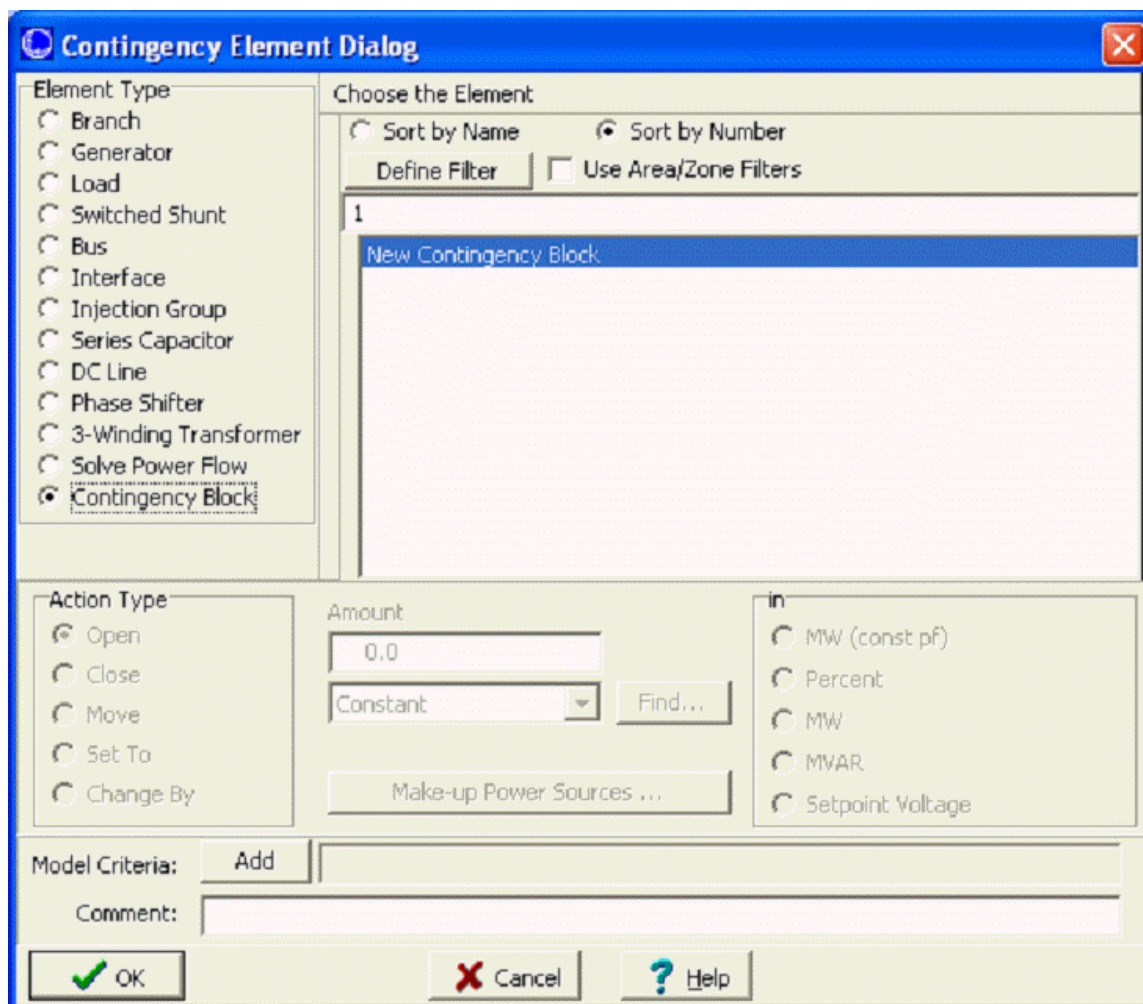
Contingency blocks are defined by clicking the **Contingency Blocks and Global Actions** button on [Advanced Modeling tab](#) on the Options page of the Contingency Analysis Dialog. Click on the Contingency Blocks tab of the Contingency Blocks and Global Actions dialog to show a display for defining Contingency Blocks. Right-clicking in the Contingency Blocks grid and choosing **Insert** allows you to insert a new Contingency Block. A dialog very similar to the [Contingency Definition Dialog](#) will open. Use this dialog to insert elements into the contingency block, create additional contingency blocks, or modify existing contingency blocks. Once a new contingency block has been created, actions can be added to the block by right-clicking in the Contingency Definition grid and choosing **Insert**. This will open the [Contingency Element Dialog](#) that allows the addition and modification of elements in the contingency block. When using the Contingency Element Dialog with Contingency Blocks, the option to insert an Element Type of *Contingency Block* is not available.

The format of the string that describes the contingency block actions and the **Model Criteria** and **Status** fields are set in the same manner as described in the [Contingency Definition Display](#).

To disable the use of a Contingency Block, set the **Skip** field to *YES*. When a Contingency Block is disabled, all of the actions defined in the block will be ignored in all of the contingencies in which the block is included.



Once Contingency Blocks have been defined, a new Element Type, *Contingency Block* will appear on the [Contingency Element Dialog](#). This is shown in the image below. You may then add a contingency block to a contingency by right-clicking on a contingency in the [Contingency Records](#) list display, choose show dialog (or insert if you are adding a new contingency record), click on Insert New Element, and choose Contingency Block from the list of element types.



Using a contingency block is an easy way to include a set of common actions in multiple contingency scenarios, without having to re-define the actions for each contingency.

---

## Contingency Analysis Dialog Overview

---

Simulator's contingency analysis tools can be accessed only from [Run Mode](#). To access this dialog go to the [Tools](#) ribbon tab, and choose **Contingency Analysis** from the [Run Mode](#) ribbon group. This will open the Contingency Analysis Dialog.

Each tab of the contingency analysis dialog covers a particular aspect of the analysis.

The [Contingencies Tab](#) is used primarily to manage the contingency list and to learn basic information regarding each contingency and the violations that it causes.

The [Lines, Buses, Interfaces Tab](#) is similar to the Contingencies Tab. The Contingencies Tab lists each contingency and shows which power system element violations occur under each contingency. The Lines, Buses, Interfaces Tab lists each power system element and shows which contingencies cause violations on those elements.

The [Options Tab](#) enables you to dictate various parameters for the analysis that govern such things as how violations are flagged in both the Base Case and for contingency conditions and what information should be included in the contingency report.

The [Summary Tab](#) chronicles the performance of the contingency analysis

The bottom panel of the dialog houses a row of buttons whose function at any given time depends on which of the four tabs is visible. The bottom panel also houses a text status indicator to keep you abreast of the current state of the analysis.

---

## Running the Contingency Analysis

---

To run the contingency analysis means to model and solve one or more contingencies from the case's current contingency list. Simulator's [Contingency Analysis Dialog](#) gives you several options for running the contingency analysis. You may

- Run every contingency in the contingency list (except, of course, for those you have designated to skip using the [Contingency Records Display](#)).
- Run a selected contingency to identify its limit violations and then leave the system in this post-contingency state. (Realize that prior to solving another contingency, Simulator will reset the system state to the reference state. For more information see [Contingency Case References](#)).
- Run a selected contingency to identify its limit violations and keep the resulting case as the new reference point for further contingency analysis runs.

To run the complete contingency list, do any one of the following:

- Right-click on the [Contingency Records Display](#) to bring up its [local menu](#) and select **Run Contingency Analysis**.
- Click the **Start Run** button visible from the [Contingency Tab](#).
- Click the **Start** button visible from the [Summary Tab](#).

To pause the contingency run once it has started, click either **Pause Run** on the [Contingency Tab](#) or **Pause** on the [Summary Tab](#). To resume a paused contingency run, click the **Continue** buttons on either the [Contingency Tab](#) or the [Summary Tab](#). Finally, to terminate a contingency run, click the **Abort** button on the [Summary Tab](#). The status indicator will inform you of the run's current state.

To solve a single contingency, identify its violations, and then leave the system in this post-contingency, select the contingency you wish to model in the [Contingency Records Display](#), right-click to invoke the display's [local menu](#), and select **Solve Selected Contingency**. (Realize that prior to solving another contingency, Simulator will reset the system state to the reference state).

To solve a single contingency and set it as the reference (starting) case for further contingency analysis activity, select the contingency you wish to model in the [Contingency Records Display](#), right-click to invoke the display's [local menu](#), and select **Solve and Set As Reference**. Simulator will model the selected contingency, flag its violations, and leave the resulting case in memory so your future work will affect the post-contingency system rather than the original pre-contingency state. See [Contingency Case References](#) for more information.

---

## Other Contingency Actions

---

By clicking the **Other** > button on either the [Contingency Tab](#) or the [Lines, Buses, Interfaces Tab](#) of the [Contingency Analysis Dialog](#), you have access to several actions.

### Delete All Contingencies

Click this to delete all the contingencies presently stored in memory.

### Clear All Contingency Results

Click this to clear all the results of the contingencies from memory. This will not delete the contingencies.

### Set As Reference

Click this to set the case presently in memory as the Reference State for contingency analysis. For more information on the reference state, see [Contingency Case References](#).

### Restore Reference

Click this to set the state of the present power system case back to the contingency analysis reference state. For more information on the reference state, see [Contingency Case References](#).

### Combined Tables

#### Contingency Violation Matrices

Clicking **Process Contingency Results** will go through the existing contingency analysis tool results and build tables showing the limit violation values for all violated lines, interfaces, and buses and the contingencies for which each element is violated. Upon opening, it is possible to limit processing to branches, buses, and/or interfaces. Also, you can specify whether to display percentages or actual flows in the resulting tables.

#### Contingency Violation List

Contingency Violation List lists all violations from the current contingency set in a tabular format. This lists eliminates possible clutter by not listing those contingencies for which there are no violations.

#### Tables

You can export the following tables to either an Excel spreadsheet or the clipboard.

##### Contingency Violation Table

Choose this option to create a table containing every violation found. The table will have the contingency name as the first column. The rest of the columns are determined by the columns presently being displayed in the [Contingency Violations Display](#) on the Contingencies Tab.

##### Contingency Definition Table

Choose this option to create a table containing every contingency definition. The table will have the contingency name as the first column. The rest of the columns are determined by the columns presently being displayed in the [Contingency Definition Display](#) on the Contingencies Tab.

### Produce Report

To produce a detailed report of the results of the contingency analysis, click **Produce Report**. This will launch a save window that will save the information you customized on the [Contingency Analysis Options: Report Writing](#) page. You will also be given the option of viewing the report in WordPad immediately after creating the file.



### Compare Two Lists of Contingency Results

Click this to open a dialog which allows you to specify two sets of contingency analysis results to compare. For more information on this comparison, see [Comparing Contingency Analysis Results](#).

### Filter Results Using Limit Monitoring Settings

Click this to filter the contingency analysis results using the present [Limit Monitoring Settings](#). This action will go through each violation for each contingency and verify that the element that was violated is set to be monitored. If the element is not set to be monitored, then Simulator will flag the violation internally as **inactive**. You will then not be able to see these violations on any of the displays, although they will still be saved in memory until you delete these contingencies, or reprocess them. These violations will also not be saved when you choose to save the contingency results.

Because the violations are saved in memory however, you can get them back without reprocessing the contingency list. To do this, change your Limit Monitoring Settings so that those violations will be set for monitoring again. Then click this option again.

When using this option, keep in mind that Simulator cannot filter results that do not exist in memory. If the filter is to be used to filter out different percentages of overload, then the contingency analysis should be run with the lowest percentage desired. For example, if the filter option is to be used to give all of those elements that are loaded above 70% of their limit and then those that are loaded above 90% of their limit, the contingency analysis should be run with the Limit Monitoring Settings defined to monitor elements that are loaded at 70% of their limit. Also, keep in mind that the more elements are monitored the slower the processing will be.

### Auto-fill Blank Contingency Element Comments

Selecting this option will fill the Comment field of the Contingency actions in the [Contingency Definition](#) table with a copy of the contingency action definition. The action description itself cannot be modified, but the comment can be modified to be more descriptive of the action being taken, for your own reference.

### Sensitivity Calculations >

Simulator provides the [PTDF tool](#) for calculating the impact of a MW transfer on all the transmission lines in the system. The Simulator [ATC tool](#) further extends the linearized methods by integrating linearized contingency analysis with the PTDF calculations. The sensitivity calculations provided here are an extension of this. They allow you to ask the question, **How will each contingency-caused branch or interface violation be affected by a MW transfer?**. Note: this calculation is not relevant for bus violations.

### Calculate OTDFs using existing PTDFs

Before executing this, you must first go to the [PTDF Dialog](#) and calculate the PTDFs for the transfer direction you are interested in. These PTDF values will then be used throughout the OTDF calculation. Click this to calculate OTDFs for each contingency-caused branch or interface violation. The values calculated will be a measure of what percent of a transfer would appear on the branch or interface after the respective contingency occurs. Realize for branch violations, that the sign of the OTDF value will be relative to the direction of the MW flow found during the contingency analysis (see the Element description on the [Contingency Violations Display](#)).

### Filter out Violations Using OTDFs

Once you have calculated the OTDFs using the existing PTDFs, you can then filter the results by selecting this option. A dialog will appear for you to enter a minimum OTDF value. All violations that have an OTDF smaller than this number will be flagged as

**inactive** and will not show up in the list of violations. See the note above regarding the **Filter Results using Limit Monitoring Settings** to better understand how inactive violations are treated.

---

## Contingency Analysis Dialog - Contingencies Tab

---

The **Contingencies Tab** of the [Contingency Analysis Dialog](#) provides tools for managing and simulating lists of contingencies. The top portion of the Contingencies Tab lists the contingency records that have been defined for the case. This table is called the Contingency Records Display. The contingency records display is a type of [case information display](#) and thus shares many of the properties and controls common to all other case information displays. By right-clicking on the display, you gain access to its [local menu](#), which offers several choices:

- **Insert** allows you to insert a new contingency record
- **Insert Special** gives the following options:
  - **Quick Insert of Single Element Contingency** which allows you to quickly specify a single element contingency via the [Contingency Element Dialog](#)
  - **Auto Insert Contingencies** which opens up the [Auto Insert Contingencies](#) dialog
  - **Merge Pairs of Selected Contingencies** which generates every possible pair of contingencies based on the selected contingencies. For instance, if you select 4 contingencies (A, B, C, D), then click **Merge Pairs of Selected Contingencies**, 6 new contingencies will be generated: A+B, A+C, A+D, B+C, B+D, and C+D.
  - **Clone Contingencies** which makes copies of each selected contingency
- **Delete** allows you to delete a particular contingency
- **Show Dialog** displays the [Contingency Definition Dialog](#) corresponding to a particular contingency
- **Run Contingency Analysis** (see [Running the Contingency Analysis](#))
- **Filter Results Using Limit Monitoring Settings** (see [Other Contingency Actions](#))
- **Solve Selected Contingency** (see [Contingency Case References - Reference State Solution Options](#))
- **Solve and Set as Reference** (see [Contingency Case References - Reference State Solution Options](#))
- **Compare Two Lists of Contingency Results** (see [Other Contingency Actions](#))
- Many other options (e.g., printing, finding, and sorting) which are characteristic of [case information displays](#)

You can also sort the display's contents by any field just by clicking on the field's heading. The default fields shown in the contingency records display are described at the bottom of this page.

As you scroll through the records in the contingency records display, you will notice that the contents of the tables that occupy the middle third of the contingency analysis dialog change. These tables are the [Contingency Definition Display](#) and the [Contingency Violations Display](#). These displays show the violations and definition for the contingency that is selected in the contingency records display at the top of the dialog. You may optionally hide the Contingency Definition Display by clicking on the X to the upper right of this display. To show the display again, click on the O to reopen it. You may change the relative width of the Contingency Definition and Violations display by moving your mouse over the line between the displays until your cursor changes. Then left click and drag to modify these widths.

When you first load a new contingency list into memory, the current status indicator along the bottom of the display will indicate the contingencies have been Initialized. During a contingency analysis run, the current status indicator may take on the values *Running*, *Paused*, *Aborted*, or *Finished*.

The contingency tab of the contingency analysis dialog offers several ways to run the contingency analysis. To start a run, you may click the **Start Run** button. Alternatively, you may choose **Run Contingency Analysis** from the [local menu](#) of the contingency records display. Once a contingency analysis run has started, you may pause it at any time by clicking

the **Pause Run** button, after which you may resume the run by clicking **Continue**. In addition to running the full set of contingencies, you may also choose to run just a single contingency. See [Running the Contingency Analysis](#) for more details.

Several other actions related to contingency analysis are also available from the Contingencies Tab. These are accessed by clicking on the **Other Actions** > button. They are described on the [Other Contingency Actions](#) page.

You may close the Contingency Analysis Dialog at any time either by clicking **Close** or by selecting **Close** from the [local menu](#) of the Contingency Records Display or the [Contingency Violations Display](#).

By default, the contingency records display presents the following fields:

**Label**

The name of the contingency.

**Skip**

Indicates whether Simulator should skip the corresponding contingency in performing the contingency analysis. If the value of the Skip field is *Yes* for a contingency, then that contingency will not be implemented when performing the contingency analysis. This is a toggleable field, which means that you can toggle its value by double-clicking the field.

**Processed**

Indicates whether the contingency has been analyzed yet as part of the current contingency run.

**Solved**

If the contingency has not yet been processed, which means that the contingency has not yet been implemented, then the value of this field is *No*. For contingencies that have been processed, the Solved field indicates whether the power flow case that resulted from the contingency could be solved to within tolerance. If the resulting power flow case could not be solved, you should investigate the contingency closely to determine if that contingency is indeed harmful to the stability of the system.

**Islanded Load**

Displays the sum of the amount of load that was islanded from the rest of the system due to the contingency. This load is inaccessible from the rest of the system.

**QV Autoplot?**

This option must be toggled to *Yes* if you intend to run a [QV Analysis](#) and automatically plot the QV curve for one or more contingency scenarios.

**Violations**

Identifies the number of violations caused by the particular contingency. This number represents the total number of contingencies (branch thermal violations + bus violations + interface violations) that were caused by the contingency. Depending on how you have configured the reporting of Base Case violations (see Base Case of the Options Tab), this number may include all, some, or none of the violations that were present in the Base Case model.

**Max Branch %**

Indicates the percentage overload of the worst-case branch violation. If there are no branch violations, this field will be blank.

**Min Volt**

Indicates the lowest bus voltage resulting from the contingency. If there are no low voltage violations, this field will be blank.

**Max Volt**

Indicates the highest bus voltage resulting from the contingency. If there are no high voltage violations, this field will be blank.

**Max Interface %**

Indicates the percentage overload of the worst-case interface violation. If there are no interface violations, this field will be blank.

---

## Contingency Violations Display

---

The Contingency Violations Display is used to list the violations that were caused by the contingency selected in the [Contingency Records Display](#).

The contingency violations display lists all of the power system elements that become violated as a result of the selected contingency. If you have selected a violation, you may click on the **Show related contingencies** button to view all contingencies that cause a violation on this power system element. Clicking this button automatically moves you to the [Lines, Buses Interface tab](#), and selects the appropriate power system element.

If the contingency selected in the Contingency Records Display resulted in no violations or has not yet been processed, the Contingency Violations Display will display the words None Defined. This display is a type of [Case Information Display](#) and thus shares many characteristics and controls common to all other case information displays. You can sort the list of violations by any field simply by clicking on that field's caption.

The **Combined Tables** button provides access to the Combined Tables options described in [Other Contingency Actions](#). In addition to these options, there is one other option for displaying **What actually occurred?** Selecting this will display a dialog box giving the details of the actions applied during the contingency and any actions that may have been skipped.

By default, the Contingency Violations Display contains the following fields:

### Category

The type of violation that occurred. If the violation is only due to the options set in [Advanced Limit Monitoring](#), then the type will start with the word *Change*.

### Element

A character string that describes the element that suffered the violation. This can either be a branch, a bus, or an interface. When the element is a branch, this string provides you with three pieces of information:

- The branch that was violated
- The terminal of the branch which had the highest loading
- The direction of the MW flow on this branch

Example 1: Jamie (22) -> Amy (33) CKT 1 at Amy (33)

This means that a branch connecting Jamie(22) to Amy (33) with circuit ID 1 is violated. The violation is at the Amy(33) terminal. The -> indicates that the MW flow on this line is from Jamie toward Amy.

Example 2: Xena (55) <- Harley (77) CKT 1 at Harley (77)

This means that a branch connecting Xena (55) to Harley (77) with circuit ID 1 is violated. The violation is at the Harley (77) terminal. The MW flow on this line is from Harley toward Xena.

### Value

Indicates the value of the violating quantity. For example, if the category of the violation is *Branch Amp* and the Value field is x, then the current on the violated element is x.

### Limit

Identifies the limit value that was violated. For example, if the category of the violation is *Branch Amp* and the Limit field is y, then the limit on the current that may flow through the element is y.

### Percent

The actual flow value for the element as a percentage of the limit.

**Area Name Assoc.**

Lists the areas with which the violated element is associated. If the element is a branch, Area Name identifies the area in which each of the branch's terminal resides. If the element is a bus, Area Name identifies the area in which the bus resides. If the element is an area-to-area interface, Area Name will identify the areas that the interface ties; otherwise, it will read *N/A*.

**Nom kV Assoc.**

Identifies the maximum voltage level associated with the violated element. If the violated element is a branch, then Nom kV lists the nominal voltage of its higher-voltage terminal. If the violated element is a bus, then Nom kV simply identifies the bus' nominal voltage. If the violated element is an interface that is made up strictly of branches, Nom kV lists the maximum nominal voltage of its terminals; otherwise, it will appear as *-9999.9*.

---

## Contingency Definition Display

---

The Contingency Definition Display lists the [Elements](#) assigned to the selected [Contingency](#). This display appears on both the [Contingency Tab](#) and the Lines, Buses, Interfaces Tab of the [Contingency Analysis Dialog](#), it also appears on the [Contingency Definition Dialog](#).

Select **Insert** from the local menu to add elements to the contingency. Right-click on a specific element in the display and select **Delete** from the local menu to remove the element from the contingency.

The contingency definition display is a type of [Case Information Display](#) and shares many characteristics and controls common to all other case information displays.

The Contingency Definition Display always contains the following fields:

### Actions

This shows a string which describes the action. You may customize the format of the string that describes the contingency actions by right-clicking on the Contingency Definition Display and choosing **Display Descriptions By**, and then choosing *Name*, *Num*, *Name/Num*, *PW File Format by Numbers*, *PW File Format by Name/kV* or *PTI File Format*.

### Model Criteria

Simulator allows you to define Model Criteria, which can consist of both [Model Conditions](#) and [Model Filters](#). These specify a criterion under which a contingency action would occur. For example, you could specify that a generation outage only occur if the pre-contingency flow on a line is higher than a specified amount. Normally, no Model Criteria will be specified, and this field will be blank. Also, note that Model Criteria can be overridden by the **Model Condition and Filter option** on the [Advanced Modeling tab of the Contingency Options](#) page. Model Criteria can be defined from the local menu (**Define Model Criteria** option) on the Contingency Definition Display, from the **Add** (Model Criteria) button on the [Contingency Element Dialog](#), or by using the **Model Conditions** and **Model Filters** buttons on the [Advanced Modeling tab of the Contingency Options](#) page.

### Status

Double click the Status field to toggle through the available options. The possibilities are:

- **Check** : The action will only be executed if the Model Criteria are true or if no Model Criteria are specified. **Check** is the default status setting.
- **Always**: The action will always be executed, regardless of the Model Criteria.
- **Never**: The action will never be executed, regardless of the Model Criteria. This allows you to disable a particular contingency action without deleting it.
- **PostCheck**: This action will be considered AFTER all **Check** and **Always** actions have been performed and the load flow solution solved. If the Model Criteria specified for the **PostCheck** action are met in the solved load flow solution (or if no Model Criteria are specified), then this action is taken and the load flow is again resolved. If the model conditions are not met, the action is skipped. This process repeats recursively for all Postcheck actions until complete. This behavior only occurs when the [Calculation Method](#) for the analysis is set to *Full Power Flow*. Because the power flow is not solved when either of the two dc methods is selected, the **PostCheck** status instead acts as a **Check** status.

### Comment

An optional user-specified data string associated with the action. For example, for an action with Model Criteria specified, you could add a sentence explaining why the action is only performed under the specified criteria. While this comment is not used by Simulator in any



way, it is saved with the contingency element when saving contingency records in [contingency auxiliary data files](#).

---

**Lines, Buses and Interfaces Tab**

---

The Lines, Buses and Interfaces Tab contains four sub-tabs: Lines/Transformers, Buses, Interfaces, and Nomogram Interfaces. The information contained on each of the sub-tabs provides an alternate method of viewing information similar to that contained on the [Contingencies Tab](#). The individual tabbed sheets show all model objects defined in the case (subject to area/zone/owner and advanced filters) whether each is associated with a specific contingency or not. The user can select any model object on its respective sheet to see how many times a violation occurred on the device during a run of a set of contingencies. When a particular device is selected that had at least one violation during the contingency run, the two pages at the bottom give the details of the analysis for the selected device.

Note: the information contained in the [Contingencies](#) and [Contingency Definition](#) sections of the Lines, Buses and Interfaces tab is object specific. The information present only pertains to contingencies that resulted in violations on the selected object.

---

**Lines, Buses and Interfaces Tab - Contingencies Section**

---

The Contingencies section gives a list of all the contingencies that caused a violation on the selected device during the analysis. This display is very similar to the [Contingency Violations Display](#). While the Contingency Violations Display shows the *elements violated under the contingency*, this display shows the *contingencies that caused the violation*. If you then select one of the contingencies in this list, the [Contingency Definition section](#) displays the actions that took place during the selected contingency. Also, when you have selected a contingency from this list, you can click the **Show Other Violations** button. This will change your dialog to the contingencies tab and select the contingency you have selected, thus allow you to see other violations caused by this contingency.

The **Combined Tables** button provides access to the Combined Tables options described in [Other Contingency Actions](#). In addition to these options, there is one other option for displaying **What actually occurred?** Selecting this will display a dialog box giving the details of the actions applied during the contingency and any actions that may have been skipped.

This page provides an easier way to check the contingency results when you are concerned with the results for a specific device in the system. The Lines, Buses and Interfaces Tab provides a much easier tool for this kind of examination, as opposed to looking through each contingency on the Contingencies Tab and trying to find each instance of a violation on the desired element.

---

## Contingency Definition Display

---

The Contingency Definition Display lists the [Elements](#) assigned to the selected [Contingency](#). This display appears on both the [Contingency Tab](#) and the Lines, Buses, Interfaces Tab of the [Contingency Analysis Dialog](#), it also appears on the [Contingency Definition Dialog](#).

Select **Insert** from the local menu to add elements to the contingency. Right-click on a specific element in the display and select **Delete** from the local menu to remove the element from the contingency.

The contingency definition display is a type of [Case Information Display](#) and shares many characteristics and controls common to all other case information displays.

The Contingency Definition Display always contains the following fields:

### Actions

This shows a string which describes the action. You may customize the format of the string that describes the contingency actions by right-clicking on the Contingency Definition Display and choosing **Display Descriptions By**, and then choosing *Name*, *Num*, *Name/Num*, *PW File Format by Numbers*, *PW File Format by Name/kV* or *PTI File Format*.

### Model Criteria

Simulator allows you to define Model Criteria, which can consist of both [Model Conditions](#) and [Model Filters](#). These specify a criterion under which a contingency action would occur. For example, you could specify that a generation outage only occur if the pre-contingency flow on a line is higher than a specified amount. Normally, no Model Criteria will be specified, and this field will be blank. Also, note that Model Criteria can be overridden by the **Model Condition and Filter option** on the [Advanced Modeling tab of the Contingency Options](#) page. Model Criteria can be defined from the local menu (**Define Model Criteria** option) on the Contingency Definition Display, from the **Add (Model Criteria)** button on the [Contingency Element Dialog](#), or by using the **Model Conditions** and **Model Filters** buttons on the [Advanced Modeling tab of the Contingency Options](#) page.

### Status

Double click the Status field to toggle through the available options. The possibilities are:

- **Check** : The action will only be executed if the Model Criteria are true or if no Model Criteria are specified. **Check** is the default status setting.
- **Always**: The action will always be executed, regardless of the Model Criteria.
- **Never**: The action will never be executed, regardless of the Model Criteria. This allows you to disable a particular contingency action without deleting it.
- **PostCheck**: This action will be considered AFTER all **Check** and **Always** actions have been performed and the load flow solution solved. If the Model Criteria specified for the **PostCheck** action are met in the solved load flow solution (or if no Model Criteria are specified), then this action is taken and the load flow is again resolved. If the model conditions are not met, the action is skipped. This process repeats recursively for all Postcheck actions until complete. This behavior only occurs when the [Calculation Method](#) for the analysis is set to *Full Power Flow*. Because the power flow is not solved when either of the two dc methods is selected, the **PostCheck** status instead acts as a **Check** status.

### Comment

An optional user-specified data string associated with the action. For example, for an action with Model Criteria specified, you could add a sentence explaining why the action is only performed under the specified criteria. While this comment is not used by Simulator in any

way, it is saved with the contingency element when saving contingency records in [contingency auxiliary data files](#).

---

## Contingency Options Tab

---

The Options Tab enables you to control many parameters that govern how the contingency analysis flags violations, deals with violations that appeared in the Base Case, and documents the violations in the form of a report.

The bulk of the Options Tab contains five sub-tabs, each of which concerns a different aspect of the contingency analysis. The sub-tabs are:

- [Modeling](#)
- [Advanced Limit Monitoring](#)
- [Advanced Modeling](#)
- [Report Writing](#)
- [Miscellaneous](#)

Please see the corresponding help sections for assistance with each tab. Once you have finished setting the options for the contingency analysis, click **Set Options**. If, at any time, you wish to revert to the set of options that were defined the last time you clicked **Set Options** (or to the default options if you have never clicked **Set Options**), click **Reset**.

---

**Contingency Options Tab: Modeling**


---

**Calculation Method**

The calculation method defines how the power flow is solved during the contingency analysis. By default, Simulator uses a **Full Power Flow** for each contingency. For a large set of contingencies and a large case, this can take some time to complete. The Full Power Flow option can utilize either an AC solution or DC solution depending on whether or not the Power Flow Solution Options are set to use the [DC Option](#). Alternatively, the contingency analysis can be done by calculating the linear approximation of the impact of the contingencies by choosing either the **Linearized Lossless DC** or **Linearized Lossless DC with Phase Shifters** method. These two methods will both solve a set of contingencies much faster than the full power flow. The Lossless DC methods utilize sensitivities of devices to calculate the load flow in a linear fashion. The only difference between the two lossless DC methods is that the first method treats all phase shifters as free-flowing, while the method with phase shifters will hold all in-service phase shifters at their present MW flow value.

**Limit Monitoring Settings**

Click this button to open the [Limit Monitoring Settings Dialog](#) to change the limit settings for monitored elements. See [Limit Monitoring Settings](#) for more details.

**For DC methods, allow amp limits by assuming a constant voltage magnitude**

If a Lossless DC calculation method has been selected from the [Modeling Tab](#), this option will become available. If checked, Simulator will allow converting MVA limits to Amp limits by assuming constant voltage magnitudes based on the state operating point just prior to the contingency calculations. Converting to amp limits is only enforced if the [Limit Monitoring Settings Option of Treat Line Limits as Equivalent Amps](#) is checked. When this option is checked, the contingency loading of the line will be calculated from the post-contingency current on the line and the calculated amp limit.

**Retry solution using the Robust Solution Process after a contingency solution failure**

Checking this option will force the contingency analysis routine to attempt a robust solution following the failure of a standard Newton-Raphson solution. The robust solution attempt does not guarantee convergence, but will attempt to slowly approach a convergence solution if possible. This process is described in more detail under [Simulation Control](#).

**Use specific solution options for contingencies**

When checked, Simulator will use a different set of [Solution Options](#) when solving the contingencies defined in the contingency set. To define the solution options used during the contingency analysis, click the **Define Solution Options** button to open the [Contingency Solution Options Dialog](#).

**Do Not Use Post Power Flow Solution Action List**

Checking this option prevents any globally defined post power flow solution actions defined with the case from being evaluated and performed in the post-contingency power flow solution during contingency analysis runs.

**Make-Up Power**

When solving a contingency, the make-up power defines how the post-contingency solution accounts for the change in system losses, generation, and load. There are three options for this make-up power.

Determine make-up using:

**Area Participation Factors specified below**

Simulator models this by temporarily switching to [Island-Based AGC](#) and using the **Calculate Participation Factors from Area Make Up Power Values** option during the contingency solution. The values specifying the **CTG Make Up Gen** for each area are used to determine the contribution of each area in an island to the amount of make-up power needed. Within each area, the individual participation factors of generators on AGC determine how much power will come from each generator. The [Calculate Participation Factors from Area Make Up Power Values section of Island-Based AGC](#) describes this in further detail.

**Generator Participation Factors From Entire Case Directly**

Simulator models this by temporarily switching to [Island-Based AGC](#) and using the **Use Participation Factors of Individual Generators** option during the contingency solution. The individual participation factors of each generator within each island are used to determine how each generator meets the make-up power needs of the island. Area and super area interchange (ACE) requirements are ignored, and all generators on AGC are dispatched to account for the changes in system losses, generation, and load.

**Same as Power Flow Case**

This uses the area interchange options specified in the normal power flow.



---

**Contingency Options Tab: Limit Monitoring**


---

The Advanced Limit Monitoring Tab allows you to shape how limit violations are detected and reported.

**Never report violations if...**

This section controls the reporting of violations that should NEVER be reported. Minimum changes in branch flows, voltages, and interface flows which must be met before a device is reported as violating a limit can be specified. These options will only be used if the checkbox is checked at the beginning of this section. The minimum change values may be specified for:

**Increase in line/transformer flows** – This is the minimum change in percentage points that the loading on a line/transformer must increase before the line/transformer gets reported as a violation. For example, if this value is set to 2%, line limits are being monitored at 100%, and a line has a base case loading of 99% and a post-contingency loading of 100%, the line will not get reported as a violation.

**Decrease in low bus voltage** – This is the minimum change in a bus voltage that must occur for a bus low voltage violation to be reported. For example, if this value is set to 0.05 pu, the low voltage limit on a bus is 0.90 pu, the base case voltage at the bus is 0.91 pu, and the post-contingency voltage at the bus is 0.89 pu, this bus will not be reported as a low voltage violation.

**Increase in high bus voltage** – This is the minimum change in a bus voltage that must occur for bus high voltage violation to be reported. For example, if this value is set to 0.05 pu, the high voltage limit on a bus is 1.1 pu, the base case voltage at the bus is 1.09 pu, and the post-contingency voltage at the bus is 1.11 pu, this bus will not be reported as a high voltage violation.

**Increase in interface flows** – This is the minimum change in percentage points that the loading on an interface must increase before the interface gets reported as a violation. For example, if this value is set to 2%, interfaces are being monitored at 100%, and an interface has a base case loading of 99% and a post-contingency loading of 100%, the interface will not get reported as a violation.

**Always report as a violation if...**

This section allows you to specify the minimum change in flow or voltage at which point any device meeting the minimum change requirement will ALWAYS be reported, EVEN if the actual device limit (flow or voltage) is NOT violated. In other words, these options allow the reporting of large changes in flow or voltage, even if the device's actual limit is NOT itself violated. These options will only be used if the checkbox is checked at the beginning of this section. The minimum change values may be specified for:

**Increase in line/transformer flows** – This is the minimum change in line/transformer flow in percentage points that the loading on a line/transformer must increase so that the line/transformer gets reported as a violation even if the loading does not exceed the element's limit. For example, if this value is set to 2%, line limits are being monitored at 100%, and a line has a base case loading of 50% and a post-contingency loading of 63%, this line will be reported as a violation even though the post-contingency loading does not exceed the limit.

**Decrease in low bus voltage** – This is the minimum amount that a bus voltage must decrease for a bus low voltage violation to be reported even if the resulting voltage is higher than the bus low voltage limit. For example, if this value is set to 0.05 pu, the low voltage

limit at a bus is 0.90 pu, the base case voltage at the bus is 1.0 pu, and the post-contingency voltage at the bus is 0.95 pu, this bus will be reported as a low voltage violation.

**Increase in high bus voltage** – This is the minimum amount that a bus voltage must increase for a bus high voltage violation to be reported even if the resulting voltage is lower than the bus high voltage limit. For example, if this value is set to 0.05 pu, the high voltage limit at a bus is 1.10 pu, the base case voltage at the bus is 1.0 pu, and the post-contingency voltage at the bus is 1.05 pu, this bus will be reported as a high voltage violation.

**Increase in interface flows** – This is the minimum change in interface flow in percentage points that the loading on an interface must increase so that the interface gets reported as a violation even if the loading does not exceed the interface limit. For example, if this value is set to 2%, interface limits are being monitored at 100%, and an interface has a base case loading of 75% and a post-contingency loading of 77%, this interface will be reported as a violation even though the post-contingency loading does not exceed the limit.

Caution should be used when using the **Always report...** options because this may result in a very large number of reported violations.

#### Report changes in bus dV/dQ sensitivity

Check this option to enable reporting changes in voltage to reactive power sensitivity for buses in the post-contingency solution. Set the sensitivity multiplier to the minimum change in sensitivity for reporting the value(s) under contingency. Any sensitivity that changes by the given multiple will be reported. You can also define an [advanced bus filter](#) using the Define Filter button, and choose either the newly defined bus filter or a previously existing bus filter for reporting dV/dQ sensitivity changes at only buses that meet the defined filter.

Also, regardless of the sensitivity multiple setting, any bus that meets the filter and has a negative dV/dQ will be reported as a violation.

Caution should be used when selecting this option. Calculating the dV/dQ sensitivities requires more computation time. Define a filter so that the dV/dQ sensitivities are only calculated for those buses for which it is important to calculate these values.

#### Report as a violations if a bus becomes disconnected

When checked, buses that become disconnected from the system due to a contingency will be reported as violations.

#### Re-reporting of base case violations

This section controls the reporting of Base Case violations. Because the concern of contingency analysis often is to identify those limitations that result directly from a particular outage or event, you may desire not to report all violations that were present in the Base Case with each contingency-specific set of violations. These options allow you to specify just how much of the Base Case violation information to report for each contingency. It gives you three options:

##### Do not report Base Case violations

When this option is checked, any element that was violated in the Base Case is omitted from the set of violations listed for each contingency.

##### List all Base Case violations for all contingencies

When this option is checked, all elements that were violated in the Base Case and are still violated post-contingency are included in the set of violations listed for each contingency.

##### Use these criteria

When this option is checked, only those elements that were violated in the Base Case and that meet the four criteria listed below will be listed with the contingency-specific violations. The four criteria include:

**Minimum % increase in line/transformer flows:** Only those branches that were violated in the Base Case whose flow has increased by at least this amount as a result of the contingency will be listed as contingency violations.

**Minimum per-unit decrease in low bus voltage:** Only those bus voltages that were violated in the Base Case that have decreased by at least this amount as a result of the contingency will be listed as contingency violations.

**Minimum per-unit increase in high bus voltage:** Only those bus voltages that were violated in the Base Case that have increased by at least this amount as a result of the contingency will be listed as contingency violations.

**Minimum % increase in interface flows:** Only those interfaces that were violated in the Base Case whose flow has increased by at least this amount as a result of the contingency will be listed as contingency violations.

When calculating the percentage for the Base Case violations used in the comparisons, the post-contingency rating sets are used. Comparisons made using percentage changes for branches are done using percentage points and not as a percentage of the actual change. For example, if the value for the minimum percent increase in line flow is 2%, lines are monitored at 100%, and a line is loaded at 100% in the base case and, the line must be loaded to at least 102% post-contingency to be reported.

#### **How to Monitor Voltage Changes**

Voltage changes can be monitored either directly on the amount of the bus per unit voltage, or based instead on a percentage change in per unit voltage from the base case per unit voltage values.

---

## Contingency Options Tab: Advanced Modeling

---

### Contingency Blocks and Global Actions

A [contingency block](#) stores a list of contingency actions and has a name (or label) associated with it. A contingency block can then be called from a contingency record. This allows you to define a block of common actions you wish to have processed during several different contingencies, and then assign the block to each contingency instead of constantly redefining the same actions for each contingency.

A [global action](#) is an action that will automatically be processed as part of EACH contingency. Thus if you have actions that are to be performed in every single contingency you define, you can insert it once in the global actions list, and Simulator will automatically use the defined action for every contingency it processes.

### Model Expressions

Clicking this button brings up a list of the [model expressions](#). You may then insert a model expression by right-clicking on the list and choosing Insert. This will bring up the Define [Model Custom Expressions](#) dialog. Model Expressions may be used as part of Model Conditions.

### Model Conditions

Clicking this button allows you to define [model conditions](#). Model Conditions may be used as part of the Model Criteria defined with [Contingency Elements](#).

### Model Filters

Clicking this button allows you to define [model filters](#). Model Filters may be used as part of the Model Criteria defined with [Contingency Elements](#).

### Generator Post-Contingency AGC

Clicking this button opens a dialog that allows the post-contingency AGC response of individual generators to be set. Set the **Post-CTG Prevent AGC Response** field to indicate whether or not a generator should be on AGC post-contingency. By default, generators are NOT set to prevent AGC response post-contingency. For the post-contingency participation factor field, **Post-CTG Part. Factor**, a value of *same* indicates that the generator should use the same participation factor as set with the generator record. Otherwise, set a numerical value for the post-contingency participation factor to use a factor other than the normal participation factor.

### Bus Load Throw Over Records

Clicking this button allows you to define load throw over records in the [Bus Load Throw Over Records](#) list display.

### Generator Maximum MW Responses in Post-Contingency

Clicking this button opens a list of generators, with a column, **CTG Max Response MW**, for entering the contingency maximum response.

During the post-contingency power flow solution, a user may enter a MW amount specifying the maximum amount of generator response from a generator. By default, these values are blank. If you enter a value in this column, then the generator response will be limited to this absolute MW response (response may be either an increase or a decrease).

Note: The user may also add a column called **CTG Max Response %** to the generator display. When entering data in this column, the **CTG Max Response MW** values will be set at a respective percent of the maximum MW output of the generator. If a generator's

maximum MW output is less than or equal to zero, then the Maximum Response will always be set to zero.

### Generator Line Drop and Reactive Current Compensation

Clicking this button opens a list of generators, with a column, **XLDC\_RCC**, for entering line drop and reactive current compensation data.

Line Drop and Reactive Current Compensation represent an alternative method for performing generator voltage control. While on LDC/RCC control, the generator will vary its MVAR output in a manner that maintains the bus voltage at a fictitious bus that is a user-specified electrical impedance of **XLDC\_RCC** away from the generator. This is called Line Drop Compensation when the impedance specified is positive, and Reactive Current Drop Compensation when the impedance specified is negative.

The impedance is specified by entering a value for **XLDC\_RCC** and the setpoint voltage is the same as used when regulating a generator in the more traditional manner. A generator will perform LDC/RCC control when it meets the following conditions:

- AVR = YES
- Use LDC\_RCC = YES

Please note that if any generators at a bus are set to **Use LDC\_RCC**, then this action will disable all traditional AVR control for generators at that bus. Other generators operating on LDC/RCC control are allowed, but no traditional AVR.

As a third special setting, you may also set the field **Use LDC\_RCC** to *PostCTG* for use in contingency analysis. When a generator is set to *PostCTG*, while implementing the post-contingency power flow, the generator will change the **Use LDC\_RCC** value to *YES*, thereby activating this new voltage control method for the generator. After the reference state is restored in the contingency analysis however, the generator will return back to a setting of *PostCTG*.

### Post-Contingency Auxiliary File

The auxiliary file specified here is loaded at the start of each contingency. In this way, very specialized post-contingency settings can be specified. An example use of this feature could be changing the Generator voltage setpoints or AVR status for the post-contingency solution or changing generator AGC status for the post-contingency.

Note: Only data stored with the contingency reference state will be "reset" when the reference state is restored. Therefore, only data stored with the reference state should be loaded via a post-contingency auxiliary file. Click [here](#) for details on the specific information stored with the reference state.

### Model reactive power for DC methods by...

When you choose to use one of the Lossless DC calculation methods, you can also specify how to handle changes in reactive power during the calculations. The lossless DC methods are based on the real power MW in the system, thus an assumption needs to be made about the reaction of the Mvar flows during the linear calculations.

The choices are:

#### Ignoring reactive power

Reactive power is completely ignored. This results in the MW flow only being compared to the limits of the elements in the system. It is important to recognize this fact, as branch and transformer limits are usually given in complex power (MVA) ratings, thus, ignoring reactive power results in comparing active power flow (MW) to total complex power limits (MVA.).

#### Assuming constant voltage magnitude

One way to include reactive power in the linearized DC results is to assume the voltage magnitudes remain constant during the linearized DC contingency analysis. Thus, the MW flows are determined from the linearized calculations, and the MVAR flows are calculated from the resulting flows and constant voltage magnitudes. Thus you will still receive an approximate complex power flow on each element (MVA), which can then be directly compared to the complex power limit of the element.

**Assuming reactive power does not change**

Another way to include reactive power in the linearized DC results is to assume that the reactive power magnitudes remain constant during the linearized analysis. Thus, the MW flows are determined from the linearized calculations, and the complex power flow of each element can be approximated using the calculated MW flows and the assumed constant Mvar flows from the base case. This again allows the approximate complex flow on each element to be compared to the complex power limit of the element.

**Model Condition and Filter options**

This section allows you to choose how assumptions about [Model Conditions](#) and [Model Filters](#) are made during the contingency analysis. Assumptions about the Model Conditions and Model Filters will impact how the [Model Criteria](#) are evaluated for contingency elements. You can specify to **Verify Model Conditions and Filters**, **Assume Model Conditions and Filters are TRUE**, or **Assume Model Conditions and Filters are FALSE**.

## Model Expressions

Model Expressions can be accessed and created by choosing **Case Information and Auxiliary > Custom Model Expressions** in the [Model Explorer](#). To open the Define Model Custom Expressions dialog, right-click and choose **Insert**, when creating a new expression, or **Show Dialog** to view an existing expression.

The **Save**, **Save As**, **Rename**, and **Delete** buttons and the drop-down list of expressions at the top of the dialog allow for the additional of new expressions, modification of existing expressions, and removal of existing expressions. There are two types of Model Expressions: an expression and a lookup table.

### Lookup Tables

The dialog as configured when creating a lookup table appears as follows:

**Define Model Custom Expressions**

Custom Expression: **Example Lookup Table**

Buttons: **Save** **Save As** **Rename** **Delete**

Choose the Type of Expression

Expression | **Lookup Table**

Lookup Type

☒ 1D Lookup ☐ 2D Lookup

Number of x1 Points: **4**

Number of x2 Points: **1**

x1 = Define... **Branch 103 (103) TO 105 (105) CKT 1 : flow as percent of limit**

x2 = Define... **Choose a Model Field**

x1	Value
50.0000	1.0000
60.0000	2.0000
75.0000	3.0000
90.0000	4.0000

Buttons: **OK** **Help** **Cancel**

A lookup table may be either one or two-dimensional by specifying the **Lookup Type**. The **Number of Points** must be set for each dimension. This will determine the size of the table. A Model Field must also be selected for each dimension. Clicking the **Define** button

next to either the **x1** or **x2** field opens the Model Field Dialog. This dialog allows the selection of an **Element Type** and a specific element of this type. The list of elements behaves the same as the [Find Dialogs](#) found throughout Simulator. After selecting the element, choose the field associated with this element that will be used in the Lookup Table. Click **OK** to accept the element and field selected and close the dialog.

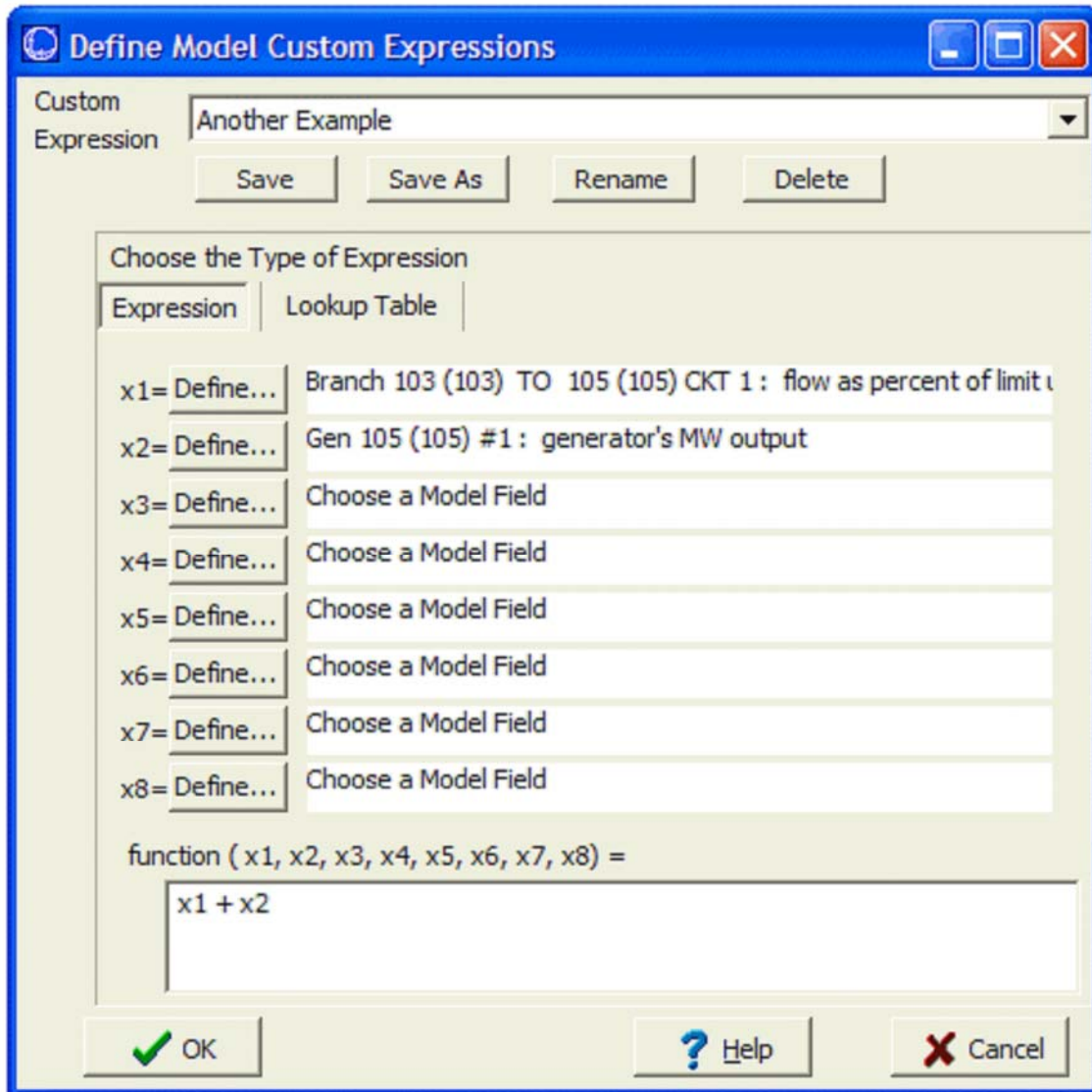
Fill in the Lookup Table with the desired values. If selecting a one-dimensional table, there will be two columns, one containing the values for the **x1** Model Field and the other containing the Values to return from the Lookup Table when the Model Field is at the defined value. There will be one header row plus as many additional rows as defined in the **Number of x1 Points**. If selecting a two-dimensional table, there will be one column for the **x1** Model Field and as many other columns as selected for the **Number of x2 Points**. There will be one header row for defining the **x2** values plus as many additional rows as defined in the **Number of x1 Points**. The cells under the first row and to the right of the first column are used for defining the Values to return from the Lookup Table when the Model Fields meet the defined values.

If selecting a one-dimensional table, the returned Value is determined based on the current value of the **x1** Model Field. The returned value is the Value corresponding with the **x1** value that is less than or equal to the current value of the Model Field. If the current value of the Model Field is less than the smallest value of **x1** in the table, then the Value returned corresponds to the smallest **x1** value. If selecting a two-dimensional table, the same rules apply except that the Value returned corresponds to the current value of the **x1** AND **x2** Model Fields and where these intersect in the table.

### Expressions

The dialog as configured when creating an expression appears below:



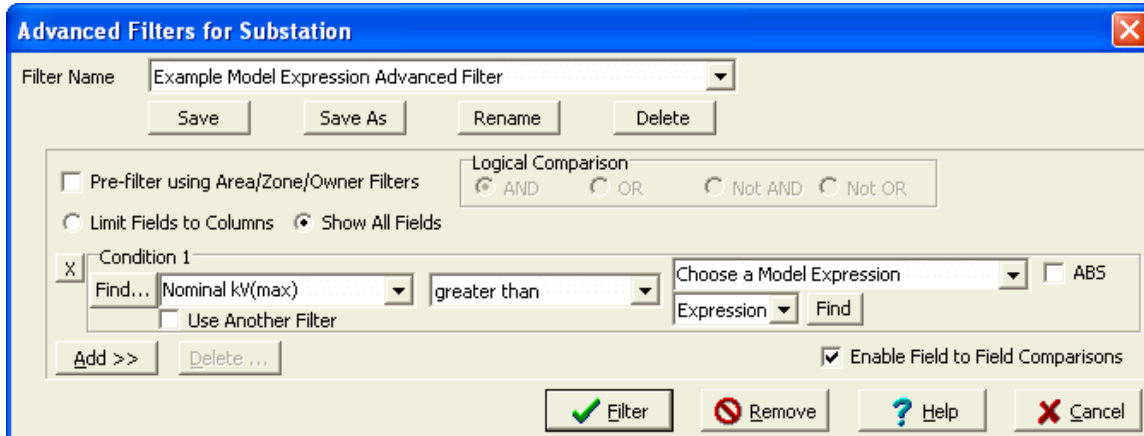


To define the Model Expression as an expression, specify the Model Fields that the expression should be a function of by clicking the **Define** button next to the appropriate variable. Clicking the **Define** button opens the Model Field Dialog. This dialog allows the selection of an **Element Type** and a specific element of this type. The list of elements of a particular type can be searched, filtered, and sorted to make finding an element easier. After selecting the element, choose the field associated with this element that will be used in the expression. Click **OK** to accept the element and field selected and close the dialog. After the Model Fields have been defined, enter the function description. See [Custom Expressions](#) for more information about defining expressions.

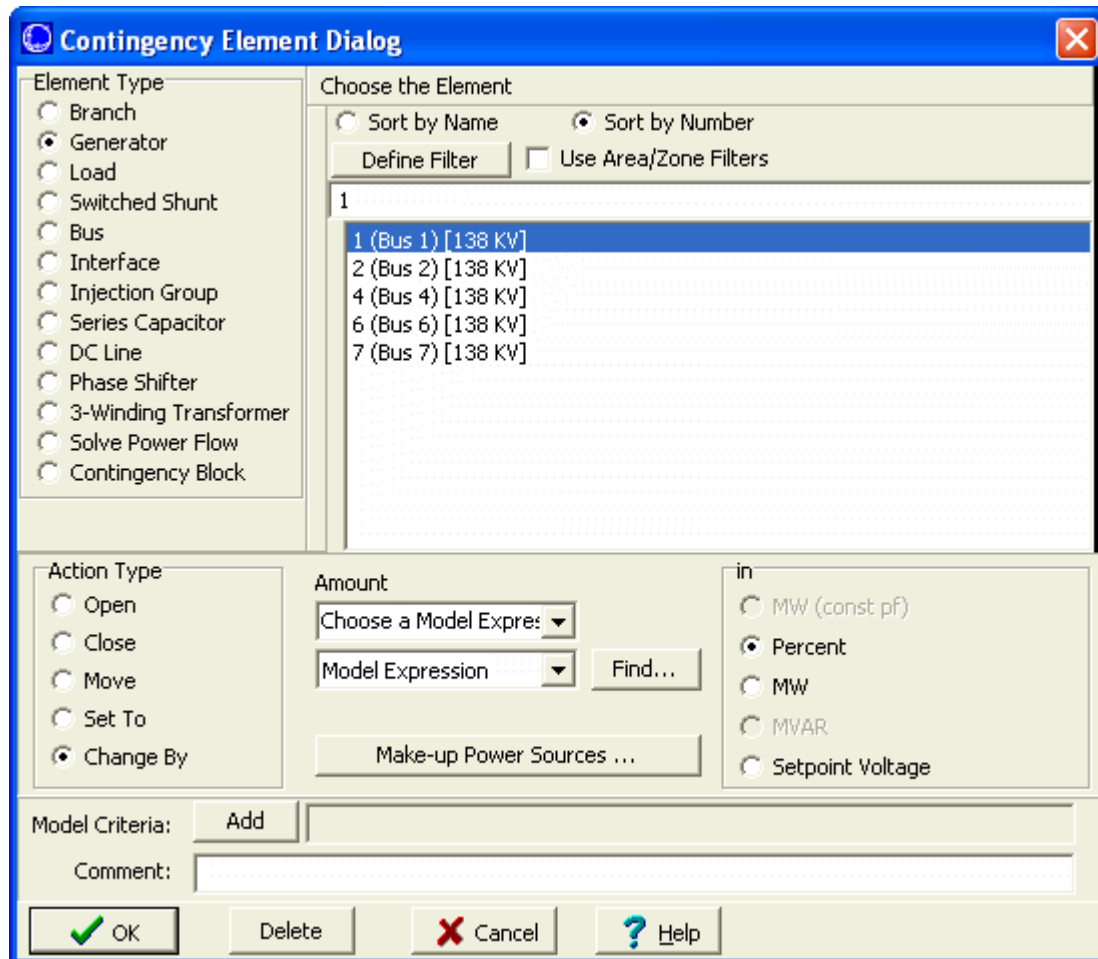
After model expressions have been defined, new options will appear on the [Contingency Element Dialog](#) and on the [Advanced Filter Dialog](#).

Normally, you may only enter a constant for the comparison value on the [Advanced Filter Dialog](#). However, if you select **Enable Field to Field Comparisons**, the advanced filter dialog will feature a drop-down from which you can choose *Expression*. After choosing

*Expression*, you may select the name of the model expression, or click the **Find** button to search for the name.



Similarly, normally you may only enter a *Constant* or a *Field* for the *Move*, *Set To*, or *Change By* **Action Types** on the [Contingency Element Dialog](#). However, now you will have the option to choose *Model Expression*. To change the type of the **Amount**, choose *Model Expression* from the second drop-down box. After doing this, enter or choose the name of the model expression in the first **Amount** drop-down box, or click on the **Find...** button to search for a name.





---

## Bus Load Throw Over Records

---

The Bus Load Throw Over list display, which is opened from the [Advanced Modeling](#) tab of the Contingency Dialog's Options page, provides you with the capability to define how load at a bus should be transferred to a different bus if the original terminal bus becomes disconnected from the system. This is referred to in Simulator as "throw over." This tool is most useful when performing contingency analysis scenarios in which buses containing loads become disconnected and you wish to analyze the impact on the system of switching the load from the disconnected bus to another bus that is still in service.

Load throw over will only attempt to move the load once, from the original bus to the load throw over bus. If the load throw over bus is already disconnected from the system, then the load will be treated as dropped during the contingency solution.

**NOTE:** The load throw over is only used when running the contingency analysis tool to analyze contingency effects on the system. Load throw over records are not used during manual solution of the power flow, even if you manually disconnect a bus with load and perform a load flow solution.

The Bus Load Throw Over display has the following fields:

**Number, Name**

The bus number and name of the load's terminal bus.

**Nom kV**

The nominal voltage level of the load's terminal bus.

**Load Throw Over Bus Number**

Enter the number of the bus you wish to have the load transferred to, should the original load terminal bus become disconnected during a contingency in the contingency analysis. The **Load Throwover Bus Name\_kV** field will be automatically populated.

**Load Throw Over Bus Name\_kV**

Enter the bus name and nominal kV (separated by a \_ between the name and nominal kV) of the bus you wish the load to be transferred to, should the original load terminal bus become disconnected during a contingency in the contingency analysis. The **Load Throw Over Bus Number** will be automatically populated if the corresponding Bus Name\_kV is found.

If choosing to identify objects by [labels](#), this field can also be used to display and enter the label of the load throw over bus.

---

## Contingency Options Tab: Report Writing

---

Simulator can produce a report that details the results of the contingency analysis (see [Contingency Analysis Summary Tab](#) to see how to generate the report). The Report Writing Tab allows you to control the content and appearance of the report. By default, the report will identify each contingency, whether or not it could be solved, and what violations resulted from it. By selecting options on this tab, you can include additional information in the report.

### Optional Report Contents

#### Case Summary

The case summary prints the [Case Description](#) and then tallies the number of different power system components in the model.

#### Option Settings

If this item is checked, the report will list each of the options selected on the [Options Tab](#) of the Contingency Analysis Dialog.

#### Monitored Areas, Monitored Zones

If either of these items is checked, the report will identify the areas and/or zones in which Simulator has looked for limit violations, and over what voltage ranges.

#### Line Flow, Interface Flow, and Bus Voltage Extremes

If one of these items is checked, the report will list the worst-case line flows or voltages seen for each monitored element during the contingency analysis. None, one, two, or all three of these can be chosen at one time.

#### Base Case Outages

If this item is checked, the report will list the limit violations that existed in the Base Case.

#### All

Selecting this option will select all of the other options.

### Identify buses by

This setting determines how the buses are listed in the data stored in the report. You can choose to have the buses displayed by number or name only, or by a combination of the number and name. You can also choose to identify with nominal voltage by checking the box labeled **Identify with Nominal Voltage**.

### Show the actions involved in each contingency

If this box is checked, the definition of each contingency will be included in the report. The definition of each contingency simply identifies the actions that were implemented as part of the contingency.

### Report only contingencies that cause violations

Checking this option will cause only the contingencies that cause violations to be shown in the report. Any contingency that did not cause a violation will not be included in the report. This option is useful if you wish to limit the size of the contingency report.

### Report only limit type with violations for each contingency

Checking this option will result in reporting only the contingency violations of the type checked in the **Limit Type Violations to Include** box.

### Report Inactive Violations and show all Rating Sets

When this box is checked, all violations that are normally being ignored during the contingency analysis (for example, base case violations) will be written to the report. Included with this option is the ability to show the different rating sets for each violated element.

#### **Limit Type Violations to Include**

Check the boxes of the types of violations you would like to be written to the report. This corresponds to the **Report only limit type with violations for each contingency** option above.

#### **Maximum Violations of a single type to report**

Enter the value of the maximum number of violations of a single type to be written to the report.

#### **Create database-friendly tables**

Checking this box will create three additional files for viewing contingency results: FILENAME\_ctgelem, FILENAME\_ctgviol, and FILENAME\_ctgstat where FILENAME is the name of the file where the main report is saved. The user can choose which symbol to use to separate the columns by choosing a delimiting symbol. These files can be easily imported into a database or spreadsheet program such as Access or Excel as delimited text files, where they can be analyzed more rigorously. These files will be automatically created and saved in the same folder as the main report.

#### **Produce Report**

Clicking this button will open a Save Dialog for selecting the file to which the report should be written. The file is saved as a text file (\*.txt). Enter the file path and name and click **Save** to write the report to the selected file. After saving the file, a dialog will provide the option of viewing the report file immediately.

---

**Contingency Options Tab: Miscellaneous**

---

**Always save results with the contingency list when you save it to a file**

If this option is checked, the violation results for each contingency will also be stored in an auxiliary file when the list of contingencies is saved to an auxiliary file. This allows for recovering the results of a contingency analysis run for a case, without reloading and re-running all the contingencies saved in the file.

**Save contingency analysis definitions/results in the case PWB file**

If this option is checked, the contingency definitions and any processed results will be saved with the load flow case in the PWB file when you save the case.

**Setting Reference Case when Contingency Analysis is Opened**

These options determine how the reference case for the contingency analysis is treated each time you re-open the contingency analysis. The first time you open the contingency analysis tool for a particular power flow case during a Simulator session, Simulator stores the current state of the power flow case as the reference case for the contingency analysis results. (See [Contingency Case References - Defining the Reference State](#) for more information.)

However, each subsequent time you open the contingency analysis dialog in the same Simulator session, Simulator needs to know how to set the reference state for the case. The choices are:

**Always set reference case to the current case**

This option will always assume that any changes you have made to the load flow case since the contingency analysis was last opened should be applied and will store the current state of the load flow as the new reference state for the contingency analysis.

**Always use the existing contingency analysis reference case**

This option will assume that the original reference case stored when the contingency analysis tool was originally launched should always be the reference state for the contingency analysis. This means that any changes that have been made to the load flow case since the initial launch of the contingency analysis will be lost, as the contingency analysis tool will reset the load flow state to the state stored with the first contingency analysis instance.

**Prompt for which reference case to use (the current case or the pre-existing reference case) whenever the Contingency Analysis Form is opened**

When this option is checked, you will always be prompted when you re-open the contingency analysis following the initial instance. You will then have the option to choose from one of the two previous settings, to either set the reference state to the current case or use the existing reference case currently stored with the contingency analysis tool.

---

### Contingency Summary Tab

---

The Summary Tab of the [Contingency Analysis Dialog](#) provides additional information on the status of the contingency analysis run and allows you to start, pause, resume, and abort the contingency run. The top half of the Summary Tab charts the progress of the contingency analysis run and issues warning messages when a particular contingency fails to solve. The next section features counters that indicate the total number of contingencies that comprise the list, the number of these contingencies that have been processed thus far, the number of contingencies that failed to solve, and the total number of violations that have been flagged. To start the contingency analysis, click **Start**. To pause the contingency analysis, click **Pause**. To resume a paused contingency run, click **Continue**. Finally, to abort the analysis, click **Abort**.

The **Refresh Displays After Each Contingency** checkbox is used to force a refresh of the counters after each contingency, which may slow down processing of the contingency set (accordingly, this box is unchecked by default to maximize solution speed).



---

## Comparing Contingency Analysis Results

---

After performing an automated contingency analysis, the results of the analysis can be saved to an auxiliary file. Once the results have been saved to a file, they can then be compared to different results in another file or to results existing in memory in the Contingency Analysis tool. To compare two sets of contingencies, do the following:

- Process each set of contingencies and save the results for at least one set in an auxiliary file.
  - o To save a contingency list, right-click on the list of contingencies and choose **Save As > Auxiliary File**.
  - o In the Save Contingency File Dialog, choose a name for the file, and then click **Save**.
  - o You will then be prompted to choose options for saving the contingency list to an auxiliary file. By default, the contingency definitions themselves are saved, along with the contingency options. The optional information you may choose to save are the [Limit Monitoring Settings](#), General [Power Flow Solution Options](#), [List Display Settings](#), and [Contingency Results](#). When saving the contingency results with the file, you may also choose to include inactive violations.  
 Inactive violations are considered violations on elements for a DIFFERENT limit than what is currently being monitored. For example, a branch may have an A limit rating of 50 MVA, and a B limit rating of 100 MVA. Consider if the B limits are being used to report violations during a contingency. A value of 75 MVA flow on the branch would not be reported in the contingency analysis as a violation, considering the B limit of 100 MVA is being used. However, Simulator internally will flag the element as a potential violation if the limit set used is switched to the A rating set. Simulator considers these types of situations as inactive violations. These are kept track of to allow the user to easily switch the rating set used for reporting violations from one set to another and see the results immediately, without having to re-run the entire contingency set to determine the violations for the new rating set. Choosing to include the inactive violations when saving an auxiliary file maintains this flexibility when the contingency definitions and results are read into a case from an auxiliary file. You can also choose for the identifiers used in the file to be either the bus numbers or the bus name and nominal kV voltage.  
 Note: For comparing two lists of contingency results, you MUST save the contingency results with each of the two auxiliary files being compared.
  - o Click **OK** to save the contingencies and the results to the auxiliary file specified.
- Once you have two different lists of contingency results (at least one of the lists must be saved in an auxiliary file), right-click on the contingency list and choose **Compare Two Lists of Contingency Results** or choose **Other > Compare Two Lists of Contingency Results**. This will bring up a dialog on which you have to specify the Contingency Lists you are interested in comparing. You must specify the **Controlling Contingency List** and the **Comparison Contingency List**. The definitions of these two lists are found below.
- Click on the **Browse** buttons to specify the two Contingency Lists you would like to compare. You can also choose to use the presently open Contingency List as either the Controlling Contingency List or the Comparison Contingency List.

### Controlling Contingency List:

The list that controls what is displayed on the dialog. Only contingencies that are defined in this list will be displayed on the form. Only violations that occur for contingencies in this list will appear in the Violations List for each contingency.

### Comparison Contingency List:

This is the list to which the Controlling Contingencies will be compared. Comparisons will occur for those contingencies in the lists that have the same **CONTINGENCY NAME**. Note that contingencies in the Comparison list whose **CONTINGENCY NAME** does not match one of those in the Controlling list will not be displayed. Also, violations which occur in a specific Comparison contingency that do not occur in the respective Controlling contingency will not be displayed.

**Example:**

A user has a power system case and a list of contingencies. The user runs contingency analysis on this system for this list of contingencies. The results are saved in a file called comparison.aux. The user now changes the system state, possibly adding in a 500 MW transaction between two areas. The contingency analysis is run on this new state of the system for the list of contingencies. The results are saved in a file called controlling.aux. You should define the contingency results you are more interested in viewing as the Controlling List because this list determines what is shown on the dialog. In this case, we are more interested in seeing the violations caused when the transaction is in place, so that list is defined as the Controlling List.

The comparison of the two sets of contingencies is now done by right-clicking and choosing **Compare Two Contingency List Results**. The file controlling.aux is set as the **Controlling List** and comparison.aux as the **Comparison List**.

- After clicking **OK** on the dialog, the contingency lists will be read from the specified files or from the presently open list. After Simulator has completed reading these files, a prompt will appear which asks, "**Would you like to set the dialog with default columns for comparing contingency lists?**". It is recommended that you choose **YES** so that the case information displays on the Contingency Dialog will automatically be set to show fields that will help you compare the two lists of contingencies. For information on the default fields used when comparing contingencies see [Comparing Contingencies List Displays](#).

---

## Comparing Contingencies List Displays

---

When comparing two sets of contingency results, there are several additional default fields added to several list displays which help you compare the results. These fields are described below for the various list displays.

### Contingencies Records on [Contingencies Tab](#)

#### Violations

The total number of violations for this contingency under the Controlling List.

#### Comp Violations

The total number of violations for this contingency under the Comparison List.

#### New Violations

The number of violations which occur in the Controlling List which do not occur in the Comparison List.

#### Max Branch % (Max Interface %)

The highest branch (interface) violation caused by this contingency in the Controlling List.

#### Comp Max Branch % (Comp Max Interface %)

The highest branch (interface) violation caused by this contingency in the Comparison List.

#### Worst Branch Violation (Worst Interface Violation)

This is the maximum of the following two values: [**Worst Increase Violation**] and [**Worst New Violation - 100%**]

#### Min Volt (Max Volt)

The worst violation in the controlling list.

#### Comp Min Volt (Comp Max Volt)

The worst violation in the comparison list.

#### Worst LowV Violation (Worst HighV Violation)

The worst new violation.

### [Contingency Violations Display](#) on [Contingencies Tab](#) and Contingency Violations on the [Lines, Buses, Interfaces Tab](#)

#### Value

The value of the violation under the controlling list.

#### Comp Value

The value of the violation under the comparison list.

#### Diff Value

The difference between **Value** and **Comp Value**

#### Limit

The limit of the element in the controlling list.

#### Comp Limit

The limit of the element in the comparison list.

**Diff Limit**

The difference between **Limit** and **Comp Limit**

**Percent**

The percent violation in the controlling list.

**Comp Percent**

The percent violation in the comparison list.

**Diff Percent**

The difference between **Percent** and **Comp Percent**

**Line/Transformer (Interface) Records on the [Lines, Buses, Interfaces Tab](#)**

**Violations**

This shows the number of branch (interface) violations which occurred in the Controlling List

**New Violations**

This shows the number of branch (interface) violations which occurred in the Controlling List, but did not occur in the Comparison List.

**Max % Loading Cont.**

The worst branch (interface) violation in the controlling list.

**Max % Ld Cont Comp**

The worst branch (interface) violation which occurred in the comparison list.

**Worst Increased Violation**

The worst increase in a branch (interface) violation from the comparison list to the controlling list.

**Bus Records on the [Lines, Buses, Interfaces Tab](#)**

**Violations**

This shows the number of violations which occurred in the Controlling List

**New Violations**

This shows the number of violations which occurred in the Controlling List, but did not occur in the Comparison List.

**Max Voltage Cont.**

The worst high voltage violation in the controlling list.

**Max Voltage Cont Comp**

The worst high voltage violation which occurred in the comparison list.

**Worst Max Volt CTG Change**

The worst increase in a high voltage violation.

**Min Voltage Cont.**

The worst low voltage violation in the controlling list.

**Min Voltage Cont Comp**

The worst low voltage violation which occurred in the comparison list.

**Worst Min Volt CTG Change**

The worst decrease in a low voltage violation.

**Nomogram Interfaces Records on the [Lines, Buses, Interfaces Tab](#)**

**Violations**

This shows the number of violations which occurred in the controlling list.

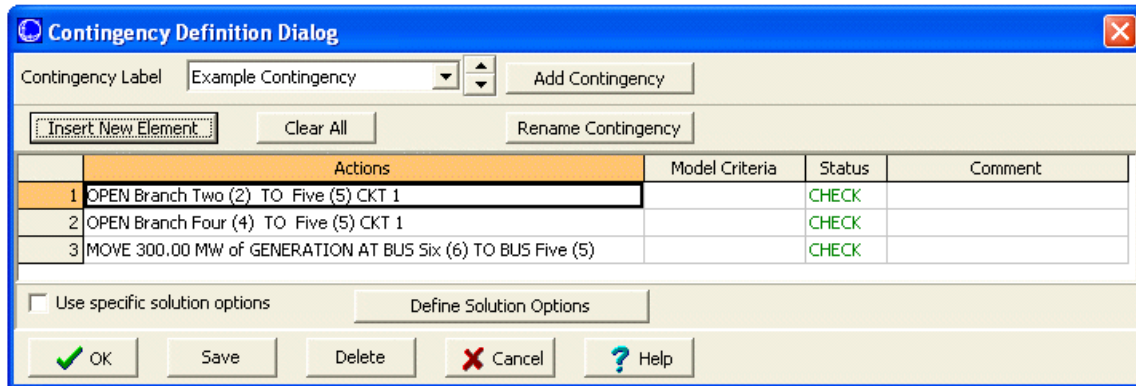
**Max % Loading Cont.**

The worst nomogram interface violation in the controlling list.

## Contingency Definition Dialog

The Contingency Definition Dialog (shown below) serves as an information source for displaying the [Contingency Element](#) (or Elements) associated with individual [Contingencies](#) defined in the case. You can use the Contingency Definition Dialog to scroll through the list of elements, to view and modify their definitions, to insert new elements in a contingency or to delete a contingency. You may access this dialog by choosing either **Show Dialog** or **Insert** from the local menu of the [Contingency Records Display](#).

After making changes, click **OK** to save your changes and close the dialog. Click **Cancel** to close the dialog without saving your changes. Click **Save** to save your changes (including the addition of a new contingency) without closing the dialog (this allows you to keep working with the dialog). Click **Delete** to remove the selected contingency from the contingency list.



### Contingency Definition Dialog

The Contingency Definition Dialog has the following controls:

#### Contingency Label

Identifies the name of the currently displayed contingency. Use the drop-down arrow to select a different contingency, or use the scroll buttons to navigate through the list of contingencies. When adding a new contingency, Contingency Label will show *New Contingency*. The user can change the Contingency Label by clicking **Rename Contingency**.

#### Add Contingency

Click the **Add Contingency** button to add a new [contingency](#) to the contingency list for the case. You will be prompted to enter a unique name for the new contingency. After naming the new contingency, the name appears in the Contingency Label and you can insert new [elements](#) in the contingency definition.

#### Insert New Element

Click this button to add a new element to the contingency. This will open the [Contingency Element Dialog](#), used to define the Action, Model Criteria and Comment associated with the element. When you return to the Contingency Definition Dialog, the display will contain the newly inserted element.

#### Clear All

Removes all elements from the contingency definition. The Contingency Elements Table will then appear blank, indicating that the contingency involves no associated actions.

**Rename Contingency**

Allows you to rename the selected contingency.

**Definitions Display**

The Contingency Definitions Display lists the [Elements](#) assigned to the selected [Contingency](#). Select **Insert** from the local menu or click on the **Insert New Element** button to add elements to the contingency. Right-click on a specific element in the display and select **Delete** from the local menu to remove the element from the contingency. For more information about this display, see the [Contingency Definitions Display](#).

**Define Solution Options**

Click this button to open the [Contingency Solution Options Dialog](#), used to define specific power flow solutions options for use under the selected contingency.

**Use Specific Solution Options**

Check this box to enable the use of Contingency Specific Solution Options (see Define Solution Options above).

## Contingency Element Dialog

The Contingency Element Dialog provides information on the individual elements that comprise a contingency definition. You may use this dialog to modify an existing contingency's definition or to add elements to new or existing contingencies.

There are several ways to open the Contingency Element Dialog:

- By pressing the **Insert New Element** button on the [Contingency Definition Dialog](#).
- By right-clicking on a [Contingency Definition Display](#) and choosing **Insert** or **Show Dialog**.
- By right-clicking on the [Contingency Tab](#) of the [Contingency Analysis Dialog](#) and choosing **Insert Special > Quick Insert of Single Element Contingency**.

After making the desired changes, click **OK** to save changes and close the dialog or click **Cancel** to close the dialog without saving your changes.

Click **Delete** to remove the element from the contingency.

Contingency Element Dialog



The Contingency Element Dialog has the following controls:

### Element Type

Indicates whether the element involved in the contingency action is a branch, generator, load, switched shunt, bus, interface, injection group, series capacitor, DC line, phase shifter, [contingency block](#) or solve power flow. The Element Type will dictate what **Action Types** are available, and the selection of the Element Type, in conjunction with the **Action Type**, **Amount**, and **in** options settings, determines what actually happens during the contingency.

What actually happens during a contingency action based on the **Element Type**, **Action Type**, and **in** options is described in detail below:

#### Branch

##### Open

The Open action will set the Status of the selected branch to *Open* if the branch Status is *Closed*. If the branch Status is already *Open*, then this action does nothing.

##### Close

The Close action will set the Status of the selected branch to *Closed* if the branch Status is *Open*. If the branch Status is already *Closed*, then this action does nothing.

#### Generator

##### Open

The Open action will set the Status of the selected generator to *Open* if the generator Status is *Closed*. If the generator Status is already *Open*, then this action does nothing.

When using the Open action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in generation due to the contingency action.

##### Close

The Close action will set the Status of the selected generator to *Closed* if the generator Status is *Open*. If the generator Status is already *Closed*, then this action does nothing.

When using the Close action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in generation due to the contingency action.

#### Move

The Move action allows the transferring of generation from an existing generator bus to another bus. A generator move is done on a bus basis and impacts the total generation of all generators at the selected bus. Generators that are adjusted by the Move action will have their AGC Status set to *No*. If there are no on-line generators at the selected generator bus, no generation move will occur. Generation may be moved in **Percent** or **MW**. A **Percent** move will move the specified percent of the current generator MW and Mvar to the defined **Bus to Move to**. A **MW** move will move the specified amount of generator MW to the Bus to Move to. The Move amount can be either positive or negative. A positive amount will decrease the output of the selected generator bus, and a negative amount will increase the output of the selected generator bus. The Bus to Move to will respond accordingly.

If a generator already exists at the bus to which the generation is being moved, the generation at that bus is adjusted according to the move amount. If a generator does not exist but a load exists, then the load is adjusted by the move amount. If no

generator or load exists, then a load is added at the bus and the injection is set according to the move amount. In terms of power injection, positive generation is the same as negative load.

### Set To

The Set To action will set the selected bus' generator parameters to a specified **Amount**. Generator Set To actions are done on a bus basis and impact the total generation of all generators at the selected bus. Generators that are adjusted by a Set To action will have their AGC Status set to *No*. If there are no on-line generators at the selected generator bus, no generation change will occur. Generator parameters can be set in **Percent**, **MW**, or **Setpoint Voltage**. When the generation is set in **Percent**, total generator MW output will be set to a value based on the specified percent of the current total generator MW output. When the generation is set in **MW**, total generator MW output will be set to the specified MW amount. When the generation is set in **Setpoint Voltage**, the voltage set-point of all generators at the selected bus will be set to the specified amount in per unit.

When using the Set To action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in generation due to the contingency action.

### Change By

The Change By action will change the selected bus' generator parameters by a specified **Amount**. Generator Change By actions are done on a bus basis and impact the total generation of all generators at the selected bus. Generators that are adjusted by a Change By action will have their AGC Status set to *No*. If there are no on-line generators at the selected generator bus, no generation change will occur. Generator parameters can be changed in **Percent**, **MW**, or **Setpoint Voltage**. When the change is in **Percent**, total generator MW output will be changed based on the specified percent of the total current generator MW output. When the change is in **MW**, total generator MW output will be changed by the specified MW amount. When the change is in **Setpoint Voltage**, the voltage set-point of all generators at the selected bus will be changed by the specified amount in per unit.

When using the Change By action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in generation due to the contingency action.

## Load

### Open

The Open action will set the Status of the selected load to *Open* if the load Status is *Closed*. If the load Status is already *Open*, then this action does nothing.

When using the Open action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in load due to the contingency action.

### Close

The Close action will set the Status of the selected load to *Closed* if the load Status is *Open*. If the load Status is already *Closed*, then this action does nothing.

When using the Close action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in load due to the contingency action.

### Move

The Move action allows the transferring of load from an existing load bus to another bus. The load move is done on a bus basis and impacts the total of all loads at the selected bus. Load may be moved in **MW (const pf)**, **Percent**, **MW**, or **Mvar**. The load amount that is moved is the total actual load and may be different than the nominal load if the area of the load has a Load MW Multiplier other than 1.0 or the load contains constant impedance or constant current components. If there is no on-line load at the selected load bus or the bus at which the load is located is *Open*, no load move will occur. A **MW (const pf)** move will move the specified amount of load MW to the **Bus to Move to** while moving a Mvar amount that will maintain a constant power factor at the selected load bus. A **Percent** move will move the specified percent of the present load MW and Mvar to the defined Bus to Move to. A **MW** move will move the specified amount of the load MW to the Bus to Move to. A **Mvar** move will move the specified amount of the load Mvar to the Bus to Move to. The Move amount can be either positive or negative. A positive amount will decrease the load at the selected bus, and a negative amount will increase the load at the selected bus. The Bus to Move to will respond accordingly.

If a load already exists at the bus to which the load is being moved, the load at that bus is adjusted according to the move amount. If no load exists, then a load is added at the bus and the load is set according to the move amount.

### Set To

The Set To action will set the selected bus' load parameters to a specified **Amount**. The load change is done on a bus basis and impacts the total of all loads at the selected bus. Load parameters can be set in **MW (const pf)**, **Percent**, **MW**, or **Mvar**. The load value that is set for each of these options is the actual constant power component of the load. This value may be different than the nominal constant power component of the load if the area that contains the load has a Load MW Multiplier other than 1.0. If there is no on-line load at the selected load bus and the bus at which the load is located is *Closed*, load status at the selected load bus will change so that one load is *Closed* and the total constant power load at the bus will be set to the specified amount. If the bus at which the load is located is *Open* or there is no existing load, either *Open* or *Closed*, at that bus, then no load change will occur. When the load is set in **MW (const pf)**, the actual constant power MW will be set to the specified amount and the actual constant power Mvar will be set so that a constant power factor is maintained at the load bus. When the load is set in **Percent**, the actual constant power MW and Mvar will be set to the specified percent of the present load MW and Mvar. When the load is set in **MW**, the actual constant power MW at the selected load will be set to the specified amount. When the load is set in **Mvar**, the actual constant power Mvar at the selected load will be set to the specified amount.

When using the Set To action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in load due to the contingency action.

### Change By

The Change By action will change the selected bus' load parameters by a specified **Amount**. The load change is done on a bus basis and impacts the total of all loads at the selected bus. Load parameters can be changed in **MW (const pf)**, **Percent**, **MW**, or **Mvar**. The load value that is changed for each of these options is the actual constant power component of the load. This value may be different than the nominal constant power component of the load if the area that contains the load has a Load MW Multiplier other than 1.0. If there is no on-line load at the selected load bus or the bus at which the load is located is *Open*, no load move will occur. When the load

is changed in **MW (const pf)**, the actual constant power MW will be changed by the specified amount and the actual constant power Mvar will be changed so that a constant power factor is maintained at the load bus. When the load is changed in **Percent**, the actual constant power MW and Mvar will be changed by the specified percent of the present load MW and Mvar. When the load is changed in **MW**, the actual constant power MW at the selected load bus will be changed by the specified amount. When the load is changed in **Mvar**, the actual constant power Mvar at the selected load bus will be changed by the specified amount.

When using the Change By action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in load due to the contingency action.

## Switched Shunt

### Open

The Open action will set the Status of the selected switched shunt to *Open* if the switched shunt Status is *Closed*. If the switched shunt Status is already *Open*, then this action does nothing.

When using the Open action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in the real power portion of the switched shunt due to the contingency action.

### Close

The Close action will set the Status of the selected switched shunt to *Closed* if the switched shunt Status is *Open*. If the switched shunt Status is already *Closed*, then this action does nothing.

When using the Close action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in the real power portion of the switched shunt due to the contingency action.

### Move

The Move action allows the transferring of switched shunt MW and/or Mvar from an existing switched shunt bus to another bus. Switched shunt moves are done on a bus basis and impact the total value of all switched shunts at the selected bus. During the move action, the nominal values of the switched shunts are adjusted, and switched shunts that are adjusted have their Control Mode set to *Fixed*. If there are no switched shunts at the selected switched shunt bus with a Status of *Closed*, no switched shunt move will occur. Switched shunt Moves may be done in **Percent**, **MW**, or **Mvar**. A switched shunt Move in **Percent** will move the specified percent of present nominal MW and Mvar at the switched shunt bus to the defined **Bus to Move to**. A **MW** move will move the specified amount of switched shunt MW to the Bus to Move to. A **Mvar** move will move the specified amount of switched shunt Mvar to the Bus to Move to. The Move amount can be either positive or negative. A positive amount will decrease the nominal MW and/or Mvar at the selected switched shunt bus, and a negative amount will increase the nominal MW and/or Mvar at the selected switched shunt bus. The Bus to Move to will respond accordingly.

If a switched shunt exists at the bus to which the switched shunt MW and/or Mvar is being moved, the switched shunt at that bus is adjusted according to the move amount. If there are existing switched shunts at the Bus to Move to but all of them have a Status of *Open*, the first switched shunt Status will be set to *Closed*, the Control Mode will be set to *Fixed*, and the nominal MW and/or Mvar will be adjusted according to the move amount. If no switched shunt exists at the Bus to Move to, then the nominal constant impedance load is adjusted according to the move

amount. If no load exists at that bus, then a load is added and the nominal constant impedance load is adjusted according to the move amount.

### Set To

The Set To action will set the selected bus' switched shunt parameters to a specified **Amount**. Switched shunt changes are done on a bus basis and impact the total nominal value of all switched shunts at the selected bus. Switched shunts that are adjusted during Set To actions have their Control Mode set to *Fixed*. Switched shunt parameters can be set in **Percent**, **MW**, **Mvar**, or **Setpoint Voltage**. When a switched shunt bus is set in **Percent**, the nominal MW and Mvar are set to the specified percent of the present nominal MW and Mvar of the switched shunt bus. When the switched shunt bus is set in **MW**, the nominal MW is set to the specified amount. When the switched shunt bus is set in **Mvar**, the nominal Mvar is set to the specified amount. When the switched shunt bus is set in **Setpoint Voltage**, the Target Value of the controllable switched shunt at the bus is set to the specified amount. Switched shunts may either regulate voltage or generator Mvar. The amount entered for the Setpoint Voltage is considered to be in per unit when the switched shunt is regulating voltage and is considered to be in Mvar when regulating generator Mvar. If there are no switched shunts at the selected bus with a Status of *Closed* when setting MW or Mvar amounts, the Status of the first switched shunt at the bus will be set to *Closed*, the Control Mode will be set to *Fixed*, and the nominal value will be set appropriately.

When using the Set To action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in switched shunt MW due to the contingency action.

### Change By

The Change By action will change the selected bus' switched shunt parameters by a specified **Amount**. Switched shunt changes are done on a bus basis and impact the total nominal value of all switched shunts at the selected bus. Switched shunts that are adjusted during the Change By action have their Control Mode set to *Fixed*. Switched shunt parameters can be changed in **Percent**, **MW**, **Mvar**, or **Setpoint Voltage**. When a switched shunt bus is changed in **Percent**, the nominal MW and Mvar are changed by the specified percent of the present nominal MW and Mvar of the switched shunt bus. When the switched shunt bus is changed in **MW**, the nominal MW is changed by the specified amount. When the switched shunt bus is changed in **Mvar**, the nominal Mvar is changed by the specified amount. When the switched shunt bus is changed in **Setpoint Voltage**, the Target Value of the controllable switched shunt at the bus is change by the specified amount. Switched shunts may either regulate voltage or generator Mvar. The amount entered for the Setpoint Voltage change is considered to be in per unit when the switched shunt is regulating voltage and is considered to be in Mvar when regulating generator Mvar. If there are no switched shunts at the selected bus with a Status of *Closed* when setting MW or Mvar amount, the Status of the first switched shunt at the bus will be set to *Closed*, the Control Mode will be set to *Fixed*, and the nominal value will be set appropriately.

When using the Change By action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for the changes in switched shunt MW due to the contingency action.

## Bus

### Open

The Open action will set the Status of all ac branches connected to the selected bus to *Open*. If all of the ac branches are already open, then this action does nothing.

## Interface

### Open

The Open action will set the Status of all ac branches in the selected interface to *Open*. If all of the ac branches are already open, then this action does nothing. The Status of any dc lines, generators, loads, or injection groups that are part of the interface will remain unchanged due to the contingency action. If the selected interface contains additional interfaces, the Status of all ac branches in the additional interfaces will also be set to *Open*.

### Close

The Close action will set the Status of all ac branches in the selected interface to *Closed*. If all of the ac branches are already closed, then this action does nothing. The Status of any dc lines, generators, loads, or injection groups that are part of the interface will remain unchanged due to the contingency action. If the selected interface contains additional interfaces, the Status of all ac branches in the additional interfaces will also be set to *Closed*.

## Injection Group

### Open

The Open action will set the Status of all generators and loads in the selected injection group to *Open* if the Status is *Closed*. If the Status is already *Open*, this action does nothing. If the injection group contains other injection groups, the Status of the generators and loads in the other injection groups will also be set to *Open*.

When using the Open action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for real power changes due to the status changes of generators and loads in the contingency action.

### Close

The Close action will set the Status of all generators and loads in the selected injection group to *Closed* if the Status is *Open*. If the Status is already *Closed*, this action does nothing. If the injection group contains other injection groups, the Status of the generators and loads in the other injection groups will also be set to *Closed*.

When using the Close action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for real power changes due to the status changes of generators and loads in the contingency action.

### Set To

The Set To action will set the real power injection of the injection group to the specified **Amount**. If loads are included in the injection group, the Mvar load will be adjusted by keeping the power factor constant. The injection can be set in **Percent**, **MW**, **Percent (In Merit Order)**, or **MW (In Merit Order)**. When setting the injection in **Percent** or **Percent (In Merit Order)**, the real power injection is set to the specified percent of the present real power injection. The present real power injection of the injection group is determined by taking the difference between the total generator MW and total load MW in the injection group. When setting the injection in **MW** or **MW (In Merit Order)**, the real power injection is set to the specified amount. When selecting either the **Percent** or **MW** option, all generators and loads in the injection group are adjusted according to their relative participation factors. All generators and loads with non-zero participation factors will be adjusted to meet the desired injection. When choosing one of the **In Merit Order** options, only generators will be adjusted to meet the desired injection. Generators will be

adjusted in order of highest relative participation factor to lowest with each generator in the list being adjusted until it hits either its maximum or minimum limit before moving on to the next generator. This process continues until the desired injection is met.

When using the Set To action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for real power changes due to the status changes of generators and loads in the contingency action.

### Change By

The Change By action will change the real power injection of the injection group to the specified **Amount**. If loads are included in the injection group, the Mvar load will be adjusted by keeping the power factor constant. The injection can be set in **Percent, MW, Percent (In Merit Order), or MW (In Merit Order)**. When changing the injection in **Percent** or **Percent (In Merit Order)**, the real power injection is changed by the specified percent of the present real power injection. The present real power injection of the injection group is determined by taking the difference between the total generator MW and total load MW in the injection group. When changing the injection in **MW** or **MW (In Merit Order)**, the real power injection is changed by the specified amount. When selecting either the **Percent** or **MW** option, all generators and loads in the injection group are adjusted according to their relative participation factors. All generators and loads with non-zero participation factors will be adjusted to meet the desired injection change. When choosing one of the **In Merit Order** options, only generators will be adjusted to meet the desired injection. Generators will be adjusted in order of highest relative participation factor to lowest with each generator in the list being adjusted until it hits either its maximum or minimum limit before moving on to the next generator. This process continues until the desired injection change is met.

When using the Change By action, the **Make-up Power Sources** button is enabled and power sources can be specified to account for real power changes due to the status changes of generators and loads in the contingency action.

### Series Capacitor

#### Bypass

The Bypass action will change the Bypass Status of the selected series capacitor to *Bypassed* if the Bypass Status is *Not Bypassed*. If the Bypass Status is already *Bypassed*, this action does nothing.

#### Inservice

The Inservice action will change the Bypass Status of the selected series capacitor to *Not Bypassed* if the Bypass Status is *Bypassed*. If the Bypass Status is already *Not Bypassed*, this action does nothing.

#### Set To

The Set To action will set the series reactance of the series capacitor to a specified **Amount**. The reactance may be set in **X (percent)** or **X (per unit)**. If the reactance of the series capacitor is set in **X (percent)**, the per unit reactance is set to the specified percent of the present per unit reactance of the series capacitor. If the reactance is set in **X (per unit)**, the per unit reactance of the series capacitor is set to the specified amount.

### DC Line

#### Open

The Open action will set the Control Mode of the selected dc line to *Blocked*. If the dc line is already blocked, then this action does nothing.

**Close**

The Close action will set the Control Mode of the selected dc line to either *Power* or *Current*. When selecting the Close action, the **Amount** must be specified for the new Setpoint of the dc line in either **MW** or **Amps**. If specifying the Setpoint in **MW**, the Control Mode of the dc line will be set to *Power*. If specifying the Setpoint in **Amps**, the Control Mode of the dc line will be set to *Current*.

**Set To**

The Set To action will set the selected dc line's Setpoint to a specified **Amount** and could possibly change the Control Mode of the line. The Setpoint can be set in **Percent**, **MW**, or **Amps**. When the dc line's Setpoint is set in **Percent**, the Setpoint is set to the specified percentage of the present Setpoint of the line and the Control Mode remains unchanged. If the Control Mode is presently *Blocked*, the resulting Setpoint will be 0 regardless of the Amount entered. When the Setpoint is set in **MW**, the Setpoint is set to the specified MW amount, and the Control Mode is changed to *Power* if the Control Mode is presently *Current*. If the Control Mode is *Blocked* or *Power*, then the Control Mode remains unchanged. When the Setpoint is set in **Amps**, the Setpoint is set to the specified current amount, and the Control Mode is set to *Current* if the Control Mode is presently *Power*. If the Control Mode is *Blocked* or *Current*, then the Control Mode remains unchanged.

**Change By**

The Change By action will change the selected dc line's Setpoint by the specified **Amount** and could possibly change the Control Mode of the line. The Setpoint can be changed in **Percent**, **MW**, or **Amps**. When the dc line's Setpoint is changed in **Percent**, the Setpoint is changed by the specified percentage of the present Setpoint of the line and the Control Mode remains unchanged. If the Control Mode is presently *Blocked*, the resulting Setpoint will be 0 regardless of the Amount entered. When the Setpoint is changed in **MW**, the Setpoint is changed by the specified MW amount, and the Control Mode is changed to *Power* if the Control Mode is presently *Current*. If the Control Mode is *Blocked* or *Power*, then the Control Mode remains unchanged. When the Setpoint is changed in **Amps**, the Setpoint is changed by the specified current amount, and the Control Mode is set to *Current* if the Control Mode is presently *Power*. If the Control Mode is *Blocked* or *Current*, then the Control Mode remains unchanged.

**When the Control Mode changes from either *Power* to *Current* or from *Current* to *Power*, the appropriate conversions are done so that the resulting Setpoint value reflects the original Setpoint and the change in Setpoint in the same units.**

**Phase Shifter****Set To**

The Set To action will set the middle of the regulation range of the selected phase shifter to the specified **Amount**. The middle of the regulation range is calculated by taking the average of the Regulation Minimum MW Flow and Regulation Maximum MW Flow. The middle of the regulation can be set in **Percent** or **MW**. If the middle of the regulation range is set in **Percent**, the middle of the range is set to the specified percent of the present middle of the regulation range. If the middle of the regulation range is set in **MW**, then the middle of the regulation range is set to the specified amount. Regardless of the method used to set the middle of the regulation range, the regulation range will remain the same, but the Regulation Minimum MW Flow and Regulation Maximum MW Flow will be adjusted so that the specified middle of the regulation range is met.



### Change By

The Change By action will change the middle of the regulation range of the selected phase shifter by the specified **Amount**. The middle of the regulation range is calculated by taking the average of the Regulation Minimum MW Flow and the Regulation Maximum MW Flow. The middle of the regulation range can be changed in **Percent** or **MW**. When the middle of the regulation range is changed in Percent, the change is based on the specified percent of the present middle of the regulation range. When the middle of the regulation range is changed in **MW**, the middle of the present regulation range is changed by the specified amount. Regardless of the method used to change the middle of the regulation range, the regulation range will remain the same, but the Regulation Minimum MW Flow and Regulation Maximum MW Flow will be adjusted so that the specified middle of the regulation range is met.

### 3-Winding Transformer

#### Open

The Open action will set the Status of the selected three-winding transformer to *Open* if the three-winding transformer Status is *Closed*. If the three-winding transformer Status is already *Open*, then this action does nothing.

#### Close

The Close action will set the Status of the selected three-winding transformer to *Closed* if the three-winding transformer Status is *Open*. If the three-winding transformer Status is already *Closed*, then this action does nothing.

### Solve Power Flow

The solve power flow type is unique in that you can include an action that forces Simulator to solve the power flow as part of the contingency. There are rare special cases of sophisticated contingency definitions this can be used for, as requested by one or more PowerWorld customers. The load flow is already generally solved for each contingency as part of the processing.

If a contingency has one or more Solve Power Flow actions, then there will be a few changes in how the contingency definition display behaves.

- Sorting of the list of actions is no longer allowed. This is because the order of the actions is now important to how the contingency is processed.
- When you right-click on the list of contingency actions there will be two new options for Move Up and Move Down. These can be used to reorder the actions.
- On the Contingency Definition Dialog, there will be up/down arrows on the right of the dialog that may be used to reorder the actions.

### Contingency Block

The individual actions of the Contingency Block are applied according to how they are defined. There are no Action Types available for a Contingency Block.

### Choose the Element

Use this portion of the dialog to choose the element involved in this action. This behaves the same as the [Advanced Find Dialogs](#) used throughout the software.

### Action Type

Defines the change specified by the contingency action. The Action Types available depend on the **Element Type** selected. Possible Action Types include: Open, Close, Move, Set To, Change By, Bypass, and Inservice. The behavior of each of these actions for the different elements is described with the **Element Types**.

### Amount

Enterable fields used to specify the quantity of change desired for the contingency element. The Amount fields are enabled when the **Action Type** is set to *Move*, *Set To*, or *Change By*. The availability of these Action Type options changes depending on the **Element Type** selected. The top field is used to define a *Constant*, *Field*, or [Model Expression](#) depending on the entry selected in the bottom drop-down box. Use the **Find** button to display a dialog listing *Fields* or *Model Expressions*. This dialog behaves the same as the [Advanced Find Dialogs](#) used throughout Simulator.

The Amount of change that occurs is based on the value of the top field entry. The top entry field is a constant, field name, or model expression name. When this entry is a field name, then the value of the change is based on the value of the selected element field when the contingency occurs. When this entry is a model expression, then the value of the change is based on the value of the selected model expression when the contingency occurs.

The Amount of change that occurs for the selected element is based on the value entered in the top field and the parameter set with the **in** options. The availability of the **in** options will change based on the Element Type selected and the Action Type selected. The availability of these options is described in detail with the **Element Types**.

### Make-up Power Sources

Power injection contingency actions result in power imbalances - typically picked up by the system slack - that may result in Power Flow Convergence Problems. Simulator provides the option of specifying Make-up Power Sources for generation, load, injection group, and switched shunt contingencies to both offset the resulting real power imbalance and provide a more realistic simulation. See [Make-up Power Sources](#) for more information.

### Model Criteria

Click the **Add** button to specify a criterion under which the contingency action will occur. Either a [Model Filter](#) or [Model Condition](#) can be selected for the Model Criteria. For example, the user can specify that a generation outage only occur if the pre-contingency flow on a line is higher than a specified amount. This condition should be specified as a Model Condition and then this condition selected as the Model Criteria. Normally, no Model Criteria will be specified for a given action and this field will be blank.

### Comment

An optional user-specified comment string associated with the action. For example, for an action with Model Criteria specified, you could add a sentence explaining why the action is only performed under the specified criteria. While this comment is not used by Simulator in any way, it is saved with the contingency element when saving contingency records in [contingency auxiliary data files](#) or with the case PWB file.

---

## Make-Up Power Sources

---

Generator, load, injection group, and switched shunt contingencies may cause imbalances between generated power and demand. The default way to handle these imbalances is to assign them to the system slack. However, this is probably unrealistic and, depending on the size of the unit or load involved, convergence problems may result.

To implement more local compensation for generation, load, injection group, and switched shunt changes, use the Make-Up Power Sources Dialog. This dialog can be accessed by from the [Contingency Element Dialog](#) by clicking the button labeled **Make-Up Power Sources**. Most of the dialog is occupied by a grid that lists bus numbers and relative contributions. This grid is a [Case Information Display](#), so its behavior should be familiar. For example, right-click on the grid to display its local menu.

We shall call those buses that must compensate for the changes caused by a generator, load, injection group, or switched shunt contingency "compensators". To insert a new compensator, select **Insert** from the grid's local menu. This opens another dialog, where you should specify the bus number of the compensator and its contribution. Specify the contribution of the compensator either as a percentage or as a fixed number of MW. If a particular contingency has multiple compensators, then the choice of basis for the contribution (either MW or percent) should be consistent for each. After identifying the compensator, click **OK**. The grid should update with your newly added compensator.

To delete an existing compensator, select it from the grid and select **Delete** from the local menu.

To determine how much each compensator contributes to the imbalance, each contribution value is normalized to the total sum of all contribution values. This allows the values to be entered in either percent or MW. Keep in mind that when entering the values in percent that the sum of all contribution values should equal 100 or the actual contribution amounts will not occur in the expected percentages.

Compensators account for the changes caused by a generator, load, injection group, or switched shunt contingency by changing either generation or load to satisfy its defined contribution. For example, suppose compensator contributions are specified as percentages, and Simulator needs to compensate for 100 MW lost in a particular generation contingency. Suppose the contingency has 4 compensators defined as follows:

Bus Number Contribution

1	20
2	30
3	10
4	40

Suppose buses 1 and 2 are load buses, and buses 3 and 4 are generators. Then bus 1's load will decrease by 20 MW, bus 2's load will decrease by 30 MW, bus 3's generation will increase by 10 MW, and bus 4's generation will increase by 40 MW. These changes will be instituted regardless of the compensators' operating limits or AGC status. In fact, any compensating units will be set off AGC to ensure that the prevailing AGC control does not distort the dictates of the contingency. Furthermore, maximum MW limits on generators will not be checked.

If a compensator has both generators and loads attached to it, the generator will take precedence. The generator will function as the compensating device, and the load will be left unchanged.

---

### **Contingency Make-Up Sources Dialog**

---

Simulator allows you to specify buses that will serve as compensators to account for changes in the balance between generation and demand caused by generator and load contingencies. This dialog allows you to define or adjust the amount of power supplied by a compensator. For more information, please see [Make-Up Power Sources](#).

---

**Tutorial: Contingency Analysis**

---

This tutorial will walk you through the basic commands necessary to insert contingencies and have Simulator automatically analyze the results. Please see [Introduction to Contingency Analysis](#) for the necessary background information regarding the capabilities and uses of the Contingency Analysis tool.

For this tutorial, we will use an existing 7-bus case.

- Open case B7SCOPF from the *Program Files/PowerWorld/Simulator/Sample Cases* directory.
- Ensure Simulator is in [Run Mode](#).
- Select **Contingency Analysis** from the [Run Mode](#) ribbon group on the [Tools](#) ribbon tab. Simulator opens the Contingency Analysis Dialog.

The next section of the tutorial discusses Defining Contingencies and provides an example of inserting a single element contingency.

**1 2 3 4 5 6 7 8 9 10 11**

**Next**

---

**Tutorial: Contingency Analysis - Page 2**

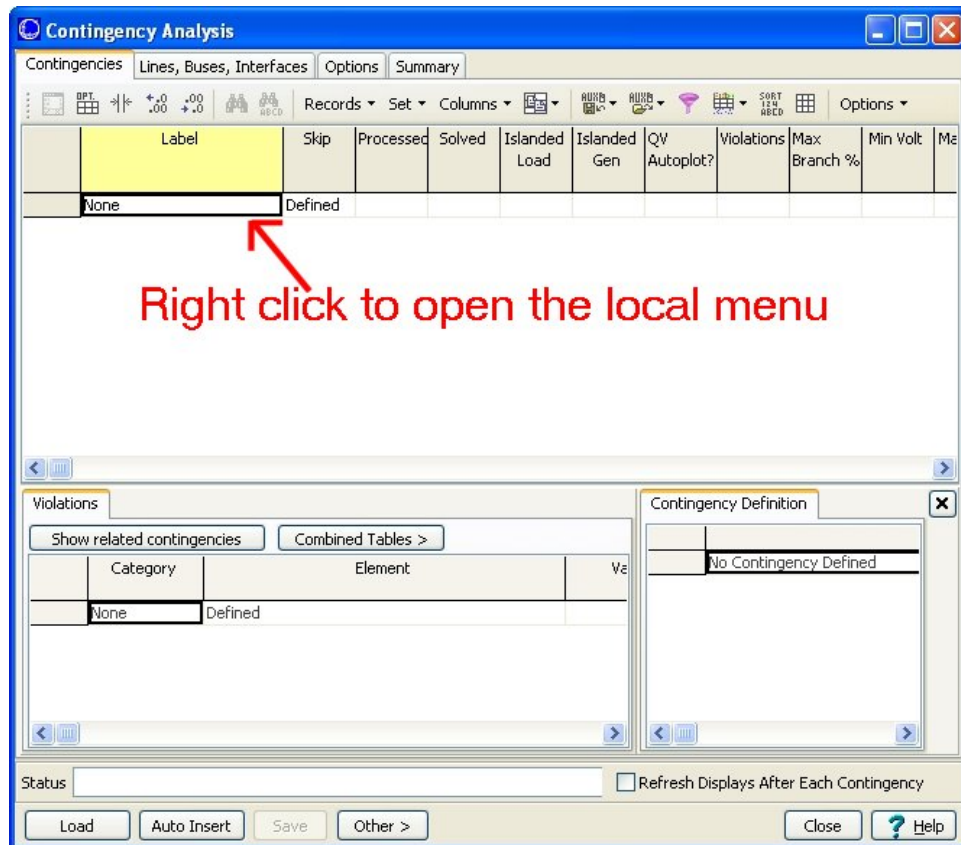

---

### Defining Contingencies

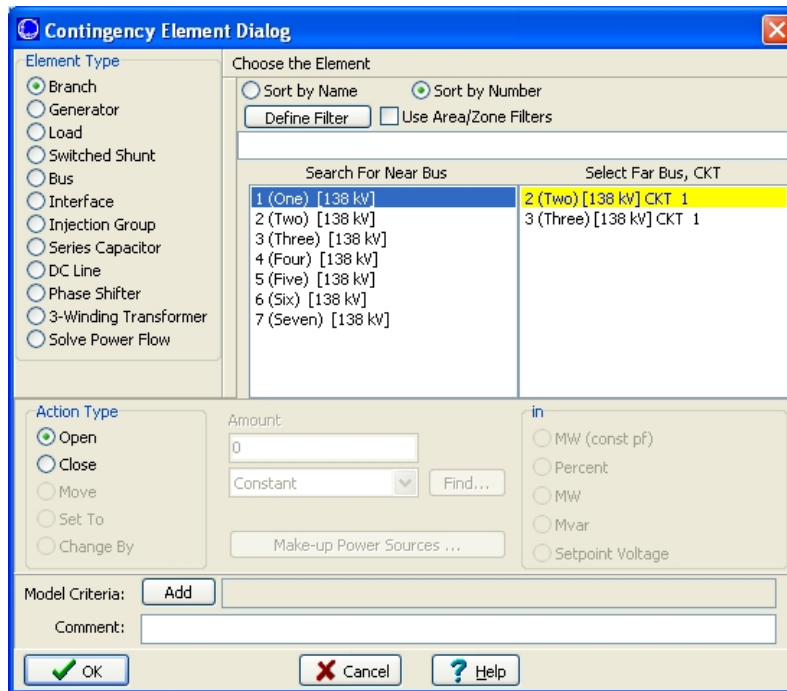
Contingencies can be either Single Element or Multiple Element. A single element contingency has only one associated contingent action. Click [here](#) for more information on the terminology used. There are four options for defining contingencies. The user may: [Load Contingencies from a File](#), [Auto Insert Contingencies](#), or use the local menu to either **Insert** contingencies or **Quick Insert of Single Element Contingency**. This tutorial will utilize the Auto Insert and Quick Insert tools.

### Quick Insert of Single Contingency Element

- Right-click on the grid (as shown below) and select **Insert Special > Quick Insert of Single Element Contingency** from the local menu. Simulator opens the [Contingency Element Dialog](#).



**Contingency Analysis Dialog**



Contingency Element Dialog

Note: The active fields in this dialog depend upon which **Element Type** is selected.

The defined action for this contingency will be to change the load at bus 2 by 25%.

- Select *Load* under **Element Type** and **2 (2) #1 [138 KV]** under **Choose the Element**.
- Select *Change By* under **Action Type**.
- Enter **25** for **Amount**, choose *Constant* from the drop-down box, then select *Percent* under the field labeled **in**.

Note that you can enter either positive or negative values in the Amount field to specify the direction that you want the change to occur (i.e. for a load contingency, entering a positive Amount increases the power consumed by the load, a negative value would decrease the load's power consumption).

Power injection contingency actions result in power imbalances - typically picked up by the system slack - that may result in Power Flow Convergence Problems. Simulator provides the option of specifying Make-up Power Sources for generation, load, injection group, and switched shunt contingencies to both offset the resulting imbalance and provide a more realistic simulation. See [Make-up Power Sources](#) for more information. See Model Criteria and Comment in the [Contingency Element Dialog](#) for more information on those fields. For the purpose of this tutorial, we will not use these fields.

- Click **OK** to insert the contingency element.

The lower right portion of the Contingencies Tab contains a Contingency Definition list display. This display lists all elements (actions) associated with the selected contingency. Since we inserted a single element contingency, there is only one element shown in the Definition.

Contingency Definition				
	Actions	Model Criteria	Status	Comment
1	CHANGE LOAD AT BUS Two (2) BY 25.00 MW (cnst pf)		CHECK	

### Contingencies Tab – Definition Display

The next portion of the tutorial provides an example of Auto Insertion of Contingency Elements.

1 2 3 4 5 6 7 8 9 10 11

Prev Next

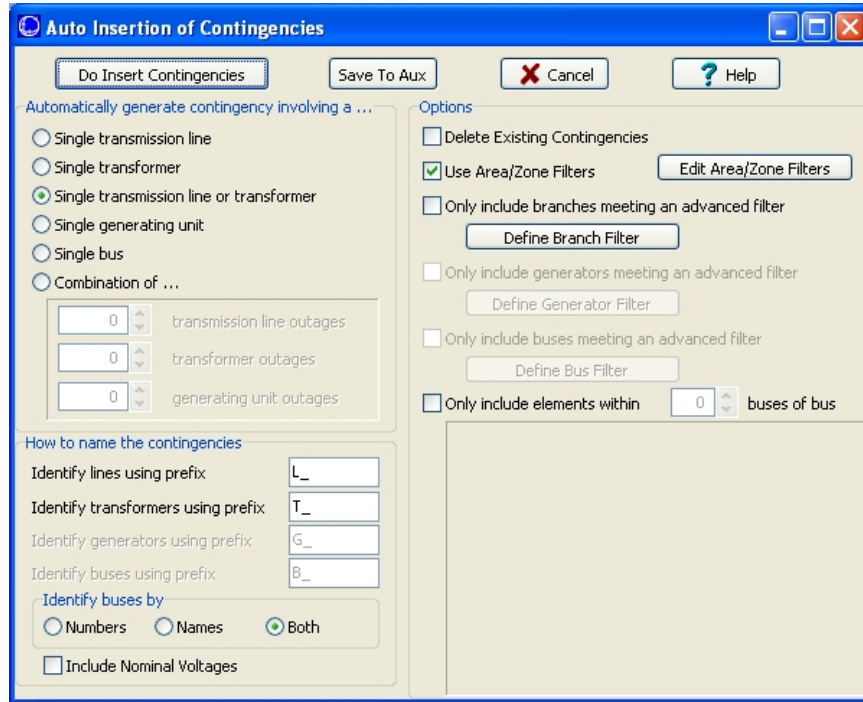


---

**Tutorial: Contingency Analysis - Page 3**


---

Simulator allows you to automatically generate a contingency list containing branch, generator and/or bus outages. To accomplish this, right-click in the Contingencies grid on the [Contingency Analysis Dialog](#), and select **Insert Special > Auto Insert Contingencies** from the popup menu. This opens the [Auto Insertion of Contingencies Dialog](#).



**Auto Insertion of Contingencies Dialog**

We will insert contingencies for all branches and generators. This will require two executions of the auto insert tool.

- Verify that **Single Transmission Line or Transformer** is selected.

When using the Auto Insert tool, you can limit the contingencies inserted to only those meeting a defined filter. We want to insert contingencies for all branches and generators so no filtering is desired.

- Click to remove the checkmark in **Use Area/Zone Filters**.
- Verify that no other options are selected.
- Click the **Do Insert Contingencies** button to accept the remaining default values and automatically insert the branch contingencies.
- Click **Yes** when asked to confirm the insertion of 11 contingencies.

Note that the Contingencies Tab of the Contingency Analysis Dialog now shows 12 contingencies.

- Right-click on the list display of the contingencies tab and select **Insert Special > Auto Insert Contingencies** from the local menu.
- Select **Single Generating Unit** then click the **Do Insert Contingencies** button. Click **Yes** to complete the auto insert of 4 generator contingencies.

Note that the Auto Insert tool did not insert a contingency for the generator connected to the Slack Bus. You can manually insert slack bus generator contingencies.

The contingencies tab now shows 16 contingency records. You can click on an individual record and view its information in the [Contingency Definition](#) section of the Contingencies Tab.

The next section of the tutorial discusses running the contingency analysis.

**1 2 3 4 5 6 7 8 9 10 11**

**Prev Next**

## Tutorial: Contingency Analysis - Page 4

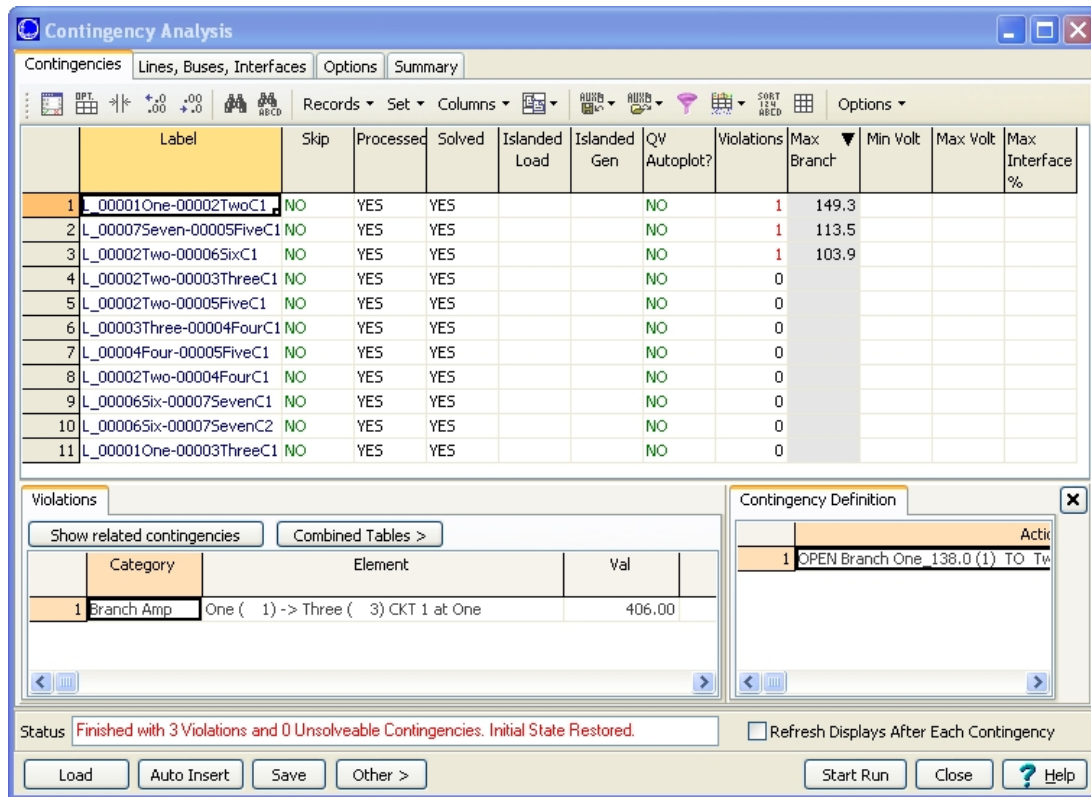
When running the Contingency Analysis, the user has three options:

1. Run every contingency on the list of contingencies (click **Start Run** on the Contingencies Tab, click **Start** on the Summary Tab or select **Run Contingency Analysis** from the local menu of the list of contingencies)
2. Run a single contingency (discussed in the advanced tutorial section) or
3. Run a single contingency then save the post-contingency state as the new reference state (also discussed in the advanced tutorial section)

We will run every contingency in the list for this portion of the tutorial.

- Press Start Run on the Contingencies Tab. Note: **Pause** and **Abort** buttons are available on the dialog while the contingency analysis is running. These may prove useful when processing a long list of contingencies.

The results from the run are shown in the Contingency Analysis Dialog.



### Contingency Analysis Dialog – Contingencies Tab

Note: The **Refresh Displays after Each Contingency** option (lower right) can slow down the analysis significantly when running a long list of contingencies.


The contingency analysis results are sorted on the contingencies tab in descending order by worst violation. We see that there were three violations resulting from the contingency analysis (one for each of the first three contingency records displayed) and that no unsolvable load flows resulted (as shown in the Status Section of the Contingencies Tab.) If you enlarge the Contingency Analysis Dialog (by dragging either side or the corner of the pane) you see

more information in the Status Window, specifically, "Finished with 3 Violations and 0 Unsolvable Contingencies. Initial State Restored."

Contingency Analysis always stores a Reference State or pre-contingency state. The Reference State stores information pertaining to: buses, switched shunts, limit groups, loads, branches, generators, areas / super areas, and power flow solution options.

Both prior to and following completion of solving a list of contingencies, the reference state is loaded into memory. This ensures that all contingency analysis solutions start from a common base case and that the system is restored to its initial state following a solution. The last sentence in the Status portion of the dialog, "Initial State Restored," tells the user that the simulation case was restored to the reference state upon completion of the Contingency Analysis Run. See [Contingency Case References](#) for more information on the Reference State.

The [Violations Section](#) (lower left) of the Contingencies Tab provides a description of each violation resulting from the execution of the contingency selected in the list of contingencies. Scroll through the list of contingencies to view information about the resulting violations for each.

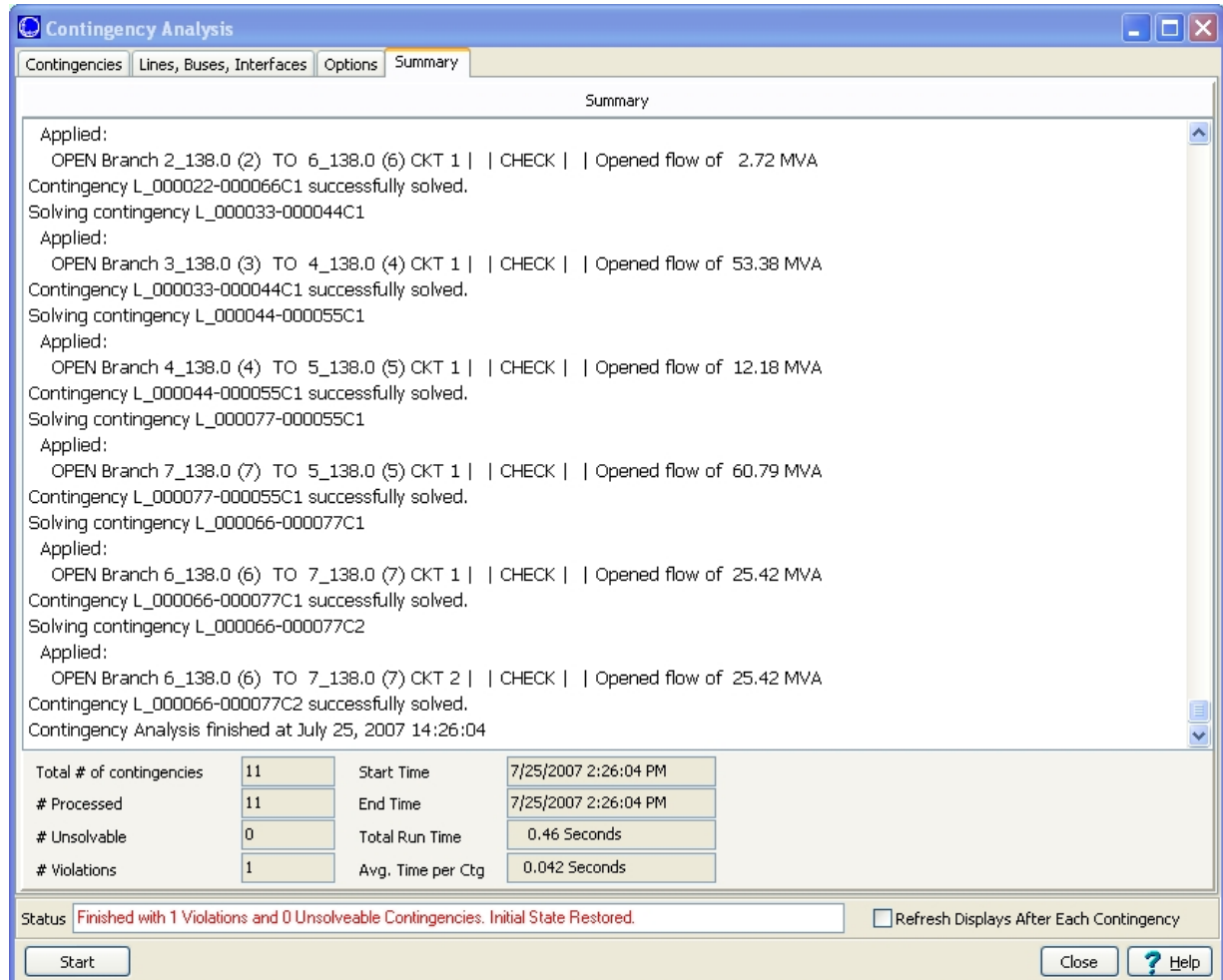
Note: You can hide the Contingency Definition section of the Contingencies Tab by clicking the  button.

**1 2 3 4 5 6 7 8 9 10 11**

**Prev Next**

## Tutorial: Contingency Analysis - Page 5

The [Summary Tab](#) of the Contingency Analysis Dialog provides the status of the present contingency analysis run.



**Contingency Analysis Dialog – Summary Tab**

1 2 3 4 5 6 7 8 9 10 11

Prev Next

## Tutorial: Contingency Analysis - Page 6

- Click on the Lines, Buses, Interfaces Tab of the dialog.

The Lines, Buses and Interfaces Tab contains four sub-tabs: Lines/Transformers, Buses, Interfaces, and Nomogram Interfaces. The information contained on each of the sub-tabs provides an alternate method of viewing information similar to that contained on the [Contingencies Tab](#). The individual tabbed sheets show all model objects defined in the case (subject to area/zone/owner filters) whether each is associated with a specific contingency or not.

**Contingency Analysis**

Contingencies Lines, Buses, Interfaces Options Summary

Records Geo Set Columns Options

Lines/Transformers Buses Interfaces Nomogram Interfaces

	From Number	From Name	To Number	To Name	Circuit	Xfrmr	Aggr Percent Overload	Violations	Max % Loading Cont.
1	1 1		3 3		1	No	31.09	1	131.09
2	1 1		2 2		1	No	28.71	1	128.71
3	2 2		5 5		1	No	6.30	1	106.30
4	2 2		3 3		1	No	0.00	0	
5	2 2		6 6		1	No	0.00	0	
6	3 3		4 4		1	No	0.00	0	
7	6 6		7 7		1	No	0.00	0	
8	7 7		5 5		1	No	0.00	0	
9	4 4		5 5		1	No	0.00	0	
10	6 6		7 7		2	No	0.00	0	
11	2 2		4 4		1	No	0.00	0	

Contingencies

Show Other Violations Combined Tables >

	RANK Overload	Label	Category	Val	Lim
1	2.2	L_000011-000022C1	Branch MVA	157.31	

Contingency Definition

	Actions
1	OPEN Branch 1_138.0 (1) TO 2_138.0 (2) C

Status: **Initialized** ☐ Refresh Displays After Each Contingency

Load Auto Insert Save Other > Start Run Close Help

### Contingency Analysis Dialog – Lines, Buses, Interfaces Tab

The user can select any model object on its respective sheet to see how many times a violation occurred on the device during the contingency analysis. When a particular device is selected that had at least one violation during the contingency run, the [Contingencies](#) and [Contingency Definition](#) sections give the details of the specific contingencies that caused the violation (or violations) on the selected device. For example, on the Lines/Transformers sub-tab:

- Select the line from 2 to 5 as shown above. This line had one violation on the most recent contingency analysis run. The contingencies section shows which contingency caused the violation and the contingency definition section details the elements that define the contingency.
- Switch back to the Contingencies Tab of the dialog.
- Scroll down in the list of contingencies and select the contingency labeled CHANGE LOAD AT BUS 2 (2)
- Right-click on the element displayed in the Contingency Definition Section of the Contingencies Tab
- Select **Show Dialog** from the local menu.
- Modify the element's **Action** to: *Set To 600 MW (const pf)* using the **Action Type**, **Amount**, and **in** fields of the Contingency Element Dialog.
- Click **OK** to close the dialog. The Contingency Definition should now show SET LOAD AT BUS 2\_138 (2) TO 600.00 MW (const pf).
- Click **Start Run** on the dialog and **Yes** when asked to confirm.
- Following the run, the Status field now shows, "Finished with 7 Violations and..." The increase in load resulted in four contingency violations not present during the last run.
- Again select the contingency labeled SET LOAD AT BUS 2... on the list display. The Violations section shows the four branch violations that occurred for the selected contingency.
- Now switch to the Lines, Buses, Interfaces tab.
- Select the line from 2 to 5. This line experienced violations under two contingencies during the run. The Contingencies section now shows the details of both contingencies (opening line 5 to 7 and changing the load at bus 2) that caused overloads on the line. Notice that the information in the Contingency Definition section is specific to the Contingency selected in the Contingencies section.

Note: The **Show Related Contingencies** and **Show Other Violations** buttons (on the Contingencies and Lines, Buses, Interfaces tabs respectively) provide a fast method of switching between the two tabs and viewing related information.

1 2 3 4 5 6 7 8 9 10 11

Prev Next

---

**Tutorial: Contingency Analysis - Page 7**

---

You now have enough information to effectively utilize PowerWorld Simulator's Contingency Analysis tool. The remaining portions of the tutorial introduce the full capabilities of the Contingency Analysis tool and provide links to the applicable Help files.

**Contingency Records**

There are four methods of defining contingencies. See [Defining Contingencies](#) for more information. Contingency Records can also be [saved to](#) or [loaded from](#) a file.

**Contingency Blocks and Global Actions**

The user may desire to have a common set of actions occur during more than one (or all) contingencies. Instead of repeatedly defining the same contingency element (or elements) in multiple contingencies, Simulator provides the option to use Contingency Blocks and/or Global Actions.

A [Contingency Block](#) is a set of contingency actions that can be defined and then called upon by individual contingency records.

[Global Actions](#) are actions defined by the user that will occur during all contingencies.

**Contingency Solution Options**

By default, the contingency analysis will use the same options as the power flow algorithm when solving each contingency. You may also override these options for all contingencies, and/or for a specific contingency. This results in the ability to set the power flow solution options in contingency analysis at three different levels

1. Contingency Specific Options (see [Contingency Definition Dialog](#))
2. Contingency Analysis Options (see [Contingency Options Tab](#))
3. General Power Flow Solution Options (see [Power Flow Solution Options](#))

When Simulator executes a particular contingency, it will first look at options specified for that contingency. Any options defined for the contingency will be used. Other options set to *use default* will look to the Contingency Analysis Options. Again, any options defined for contingency analysis will be used. Finally, options marked in the Contingency Analysis Options as *use default* will be set to the same setting as the power flow solution options.

**1 2 3 4 5 6 7 8 9 10 11**

**Prev Next**



---

**Tutorial: Contingency Analysis - Page 8**

---

**Reference State**

The [Reference State](#) (introduced on page 4 of the tutorial) can prove very useful to advanced users if they know how to exploit its capabilities. When running a list of contingencies the reference state is loaded into memory prior to executing each contingency. The system is restored to the reference state following the contingency analysis run. In addition to running an entire contingency list, the user also has the option to solve individual contingencies by right clicking on the desired contingency - on the Contingencies tab of the Contingency Analysis Dialog - and selecting either **Solve Selected Contingency** or **Solve and Set as Reference** from the local menu.

**Solve Selected Contingency** causes Simulator to first load the reference state into memory then solve the contingency. THE SYSTEM STATE IS NOT RESTORED TO THE REFERENCE STATE FOLLOWING THE SOLUTION; the system state then reflects the power system flows of the post-contingency state. The advantage of this approach is the ability to implement a contingency and then modify the system looking for possible actions that might mitigate violations caused by the contingency. Be aware; however, that prior to solving another contingency, Simulator will reset the system state to the reference state thereby removing all modifications made following the previous contingency solution. The user may also automatically restore the system state to the reference state by selecting **Other > Restore Reference** from the Contingency Analysis Dialog.

**Solve and Set As Reference** acts the same as Solve Selected Contingency with one exception. After executing the contingency, the post-contingency state is automatically set as the reference state. As a result, all subsequent contingencies will use the post-contingent state as the Reference State.

**Make-up Power Sources**

Power injection contingency actions result in power imbalances - typically picked up by the system slack - that may result in Power Flow Convergence Problems. Simulator provides the option of specifying Make-up Power Sources for generation, load, injection group, and switched shunt contingencies to both offset the resulting imbalance and provide a more realistic simulation. See [Make-up Power Sources](#) for more information.

1 2 3 4 5 6 7 8 9 10 11

Prev Next

---

**Tutorial: Contingency Analysis - Page 9**

---

**Contingency Elements**

A contingency element consists of a single Contingency Action and its associated Model Criteria, Status and Comment (optional). Multiple Contingency Elements can be defined for a single Contingency.

**Contingency Action** - Click [here](#) for a list of available contingency actions.

**Model Criteria** - Model Criteria are criteria under which the contingency action will occur and consist of both [Model Conditions](#) and [Model Filters](#).

**Status** – The status field of a contingency element can take one of four values: CHECK, ALWAYS, NEVER or POSTCHECK.

- CHECK - The action will be executed if the Model Criteria are true or if no Model Criteria are specified. CHECK is the default status setting.
- ALWAYS - The action will always be executed, regardless of the Model Criteria.
- NEVER - The action will never be executed, regardless of the Model Criteria. This allows you to disable a particular contingency action without deleting it.
- POSTCHECK – Following completion of all CHECK and ALWAYS actions, the contingency analysis tool runs the load flow solution. POSTCHECK actions are then addressed recursively until all are complete. The execution of POSTCHECK actions follows the same rules as CHECK actions with the exception of not being checked until the load flow has been solved. If the Model Criteria specified for the POSTCHECK action are met in the solved load flow solution (or if no Model Criteria are specified), then the action is taken and the load flow is again resolved. If the model conditions are not met, the action is skipped.

For more information, see [Contingency Definition Display](#).

1 2 3 4 5 6 7 8 9 10 11

Prev Next

---

**Tutorial: Contingency Analysis - Page 10**


---

**Contingency Analysis Dialog – Options Tab Overview**

The bulk of the [Options Tab](#) contains five sub-tabs, each of which concerns a different aspect of the contingency analysis. A brief description of the actions allowed via each sub-tab follows. Click on the headings below for more information on each.

**Modeling Sub-tab**

The modeling tab allows the user to:

- Define the Contingency Analysis Load Flow Calculation Method
- Specify Limit Monitoring Settings
- Instruct Simulator to retry the solution using the Robust Solution Process following a failure to converge
- Specify the use of specific solution options for contingencies
- Specify Make-up Power for the post-contingency solution

**[Advanced Limit Monitoring Sub-tab](#)**

The Advanced Limit Monitoring Tab allows you to shape how limit violations are detected and reported.

**[Advanced Modeling Sub-tab](#)**

The Advanced Modeling tab allows the user to:

- Define Contingency Blocks and Global Actions (refer to [Page 7](#) of the tutorial)
- Define Model Criteria (Expressions, Conditions and Filters)
- Specify Model Criteria Options
- Define Bus Load Throw-Over Records – BLTRs provide the capability to define how load at a bus should be transferred to a different bus if the original terminal bus becomes disconnected from the system during a contingency
- Define Generator Maximum MW Responses in Post-Contingency – allows the user to limit the absolute MW response of a generator during a contingency
- Define Generator Line Drop Compensation (LDC) and Reactive Current Compensation (RCC) – LDC and RCC controls allow the user to model the real-time control done at some real generators
- Specify a post-contingency aux file to be loaded at the start of each contingency. This allows the setting of very specialized post contingency settings.
- Specify how reactive power is modeled for DC calculation methods

**[Report Writing Sub-tab](#)**

Simulator can produce a report that details the results of the contingency. The Report Writing Tab allows you to control the content and appearance of the report.

**[Miscellaneous Sub-tab](#)**

The Miscellaneous sub-tab provides options pertaining to the loading and saving of contingency records as well as specifying how the reference state should be established when the contingency analysis tool is accessed.

**1 2 3 4 5 6 7 8 9 10 11**

**Prev Next**

---

**Tutorial: Contingency Analysis - Page 11**

---

**Contingency Analysis Dialog – Other Contingency Actions**

The **Other** > button on the Contingencies Tab and the Lines, Buses, Interfaces Tab of the Contingency Analysis Dialog, provides access to a number of additional contingency actions. Some of the actions available include:

- Deleting all contingencies
- Clearing contingency results
- Setting or restoring the reference state
- Producing combined tables of results
- Producing detailed reports of results
- Comparing lists of contingencies
- Filtering results
- Auto-filling blank comment fields and performing sensitivity calculations such as OTDFs and PTDFs.

For more information, see [Other Contingency Actions](#).

1 2 3 4 5 6 7 8 9 10 11

Prev

---

## Fault Analysis

---

Fault analysis can only be performed when Simulator is in [Run Mode](#). There are three ways to start a fault analysis study:

- Go to the [Tools](#) ribbon tab, and choose **Fault Analysis** from the [Run Mode](#) ribbon group. This is only available in [Run Mode](#).
- Right click on a bus and choose **Fault...** to perform a fault analysis at that bus
- Right click on a line and choose **Fault...** to perform a fault analysis at that point on the line

All of these options will open the [Fault Analysis](#) dialog. If you opened the dialog by right-clicking on a bus or line, the fault information on that bus or line will already be filled in. If you selected the **Fault Analysis...** option from the [Tools](#) ribbon tab, the information about the location of the fault will need to be provided.

---

## Fault Analysis Dialog

---

The **Fault Analysis** dialog can be used to perform a fault analysis study on the currently loaded power system. A fault study can only be performed while Simulator is in [run mode](#), since the load flow must be validated and solved before a fault study can be calculated. If you are observing fault analysis results in the fault analysis dialog and switch to [edit mode](#), the dialog will automatically be closed and the fault analysis results will be cleared from memory.

Simulator stores fault data in the PowerWorld binary file along with the load flow data, but by default most other load flow formats store fault data in separate files. The fault data can be stored in and loaded from an external file, but if no fault data is present in a PowerWorld binary file or loaded from an external file before a fault analysis is run, Simulator will use the load flow data as default values for the analysis. Fault data values can also be modified for specific devices by opening a specific device's information dialog and looking at the **Fault...** tab. Devices that require sequence specific data for fault analysis are [buses](#) (for sequence load injections), [generators](#), [switched shunts](#), [transmission lines](#), and [transformers](#).

Phase shifts in a fault analysis calculation can be very important for calculating the correct fault currents and voltages throughout the system. The phase shifts that are applied for transmission lines and transformers are taken from the load flow values of phase entered with each specific transmission element. While transformers can have their transformer configurations specified (i.e. Delta-Wye, Grounded Wye-Delta, etc.), these configurations are **NOT** used to determine phase shift angles, **ONLY** to determine the proper grounding on each side of the transformer. The phase shifts that are applied are taken from the load flow data phase values for the transmission elements. If no phase shifts are entered in the load flow data, the fault analysis will treat all elements as having zero phase shift. Phase shift values can be entered manually for each transmission element, but are also included in most load flow formats and will be read into Simulator when loading a load flow data file.

Note that the bus chosen for the fault is always set to a 0 degree reference, and all other buses are shifted according to this reference.

### Fault Data

The **Fault Data** tab is where the type and location of the fault are specified, and where the results of the fault analysis can be seen in tabular format.

#### Fault Location

Choose to perform the fault at a bus location, or at a point somewhere on a line. If Bus Fault is selected, the only information needed is the bus number, which needs to be entered in the Fault Bus field. If an in-line fault is desired, the from and to bus numbers, circuit ID, and location of the fault (entered in percent of total line length, measured from the From Bus) will need to be given. Selecting the **Fault...** option from the bus or line local menus will automatically set up the Fault Location fields.

#### Fault Type

Choose from one of four types of fault to calculate at the fault location:

**Single Line - to - Ground** Computes a single phase line - to - ground fault using a user defined ground fault impedance. The phase evaluated is always referenced as phase A.

**Line - to - Line** Computes a line - to - line fault, assuming an impedance of  $999 + j999$  to ground. Phases B and C are always referenced as the faulted phases.

**3 Phase Balanced** Balanced three-phase line fault - to - ground using a user-defined ground fault impedance.

**Double Line - to - Ground** Computes a line - to - line - to - ground fault, using a user defined ground fault impedance.

#### Current Units

Allows you to choose to observe the fault currents in per unit current or actual Amps.

#### Online Display

The string grids at the bottom of the **Fault Analysis** dialog will display the fault results in tabular format, but the results can also be viewed graphically on the oneline diagram by selecting an option from this group of options. Any of the three phase values can be selected for display individually, or all three phases can be viewed simultaneously. Viewing all three phases of information simultaneously can result in an abundance of information on the diagram at one time, so selective placement of the necessary bus and line fields may need to be considered when planning on viewing the fault analysis results graphically on the oneline diagram.

The fields necessary on the oneline diagram for display of the fault analysis results are:

- **Bus Voltage** and **Bus Angle** fields need to be present for a bus. When choosing to view fault analysis results, these two fields will be identified, and the actual load flow values will be replaced by the fault phase voltages (in per unit) and angles (in degrees).
- **AC Line MW Flow** and **AC Line Mvar Flow** fields need to be present for a line. These two fields will be identified, and the MW and Mvar values will be replaced by the fault phase current magnitudes (in Amps or per unit) and angles (in degrees).
- **Gen MW Output** and **Gen Mvar Output** fields need to be present for a generator. These two fields will be identified, and the MW and Mvar values will be replaced by the generator terminal fault phase current magnitudes (in Amps or per unit) and angles (in degrees).

#### Fault Current

Displays the magnitude and angle of the current at the fault location during the fault.

#### Calculate

Pressing this button will run the fault analysis. In order for the results to be calculated, the power flow has to be in a solved state for the results to have any relevance. Therefore the first thing performed when **Calculate** is pressed is to solve the power flow. You can observe this by viewing the [Message Log](#) when you run the calculation. Once the power flow has been solved, then the fault analysis calculations are run and the results displayed.

#### Clear

Pressing **Clear** will clear any fault analysis results currently in memory and displayed on the dialog.

There are also five informational displays at the bottom of this dialog for showing the fault analysis calculation results:

[Buses](#)

[Lines](#)

[Generators](#)

[Loads](#)

[Switched Shunts](#)

### Fault Options

The **Fault Options** tab is where an impedance to ground can be defined at the fault location, where fault data can be loaded from or saved to an external file, and zero-sequence mutual impedances can be viewed or changed.

### Fault Impedance

For any of the fault types calculated, a **Fault Impedance** can be included. A Resistance and Reactance can be entered as the path to ground of the fault, and is taken into account when calculating the fault current used to determine the rest of the fault values.

### Load Data / Save Data

These two buttons allow loading from and saving to external files. Currently the two types of files supported are PSS/E Sequence Data files (.seq) and [PowerWorld Simulator Auxiliary files](#) (.aux). Either one of these formats can be loaded and saved.

### Zero Sequence Mutual Impedances

Zero sequence mutual impedances can be stored and modified in the [Mutual Impedance Records](#) table. Usually the zero sequence mutual impedance parameters are read in from a sequence data file. However, it is also possible to insert and delete mutual impedance records from this table by right-clicking in the table and selecting **Insert** or **Delete** from the [local menu](#). When **Insert** or **Show Dialog** are chosen from the mutual impedance table local menu, the [Mutual Impedance Record dialog](#) will open, from which a mutual impedance record can be inserted or modified.

### Pre-Fault Profile

Changing this option determines the pre-fault voltage profile to be used for the fault analysis calculations. The pre-fault profile selection affects the sequence Y-bus values, fault currents, and post-fault voltages.

### Profile Options

Additional pre-fault profile options are available when the pre-fault profile selected is either Flat IEC-909 or Flat Classical.

**XF Turns Ratios Set to 1** If checked, all transformer tap ratios are assumed at their nominal tap position.

**Line Charging Set to 0** If checked, line charging capacitance is ignored in all calculations.

**Shunt Elements** Shunt elements (bus and line shunts) can optionally be treated normally, ignored in the positive sequence only, or ignored in all sequences.

**IEC Parameters** This option only applies to the Flat IEC-909 pre-fault profile. The pre-fault voltage magnitude can be specified for each bus. In addition, a generator power factor angle (in degrees) can be specified for use when generator currents need to be calculated based on bus voltage and power (real and reactive) delivered by the generator.

### Matrices

The **Matrices** tab is where the positive, negative and zero sequence admittance matrices can be viewed for the fault. This tab is only visible when a fault has been calculated. The three pages on the Matrices tab each have the same functionality as other [Case Information](#) displays. The purpose of each display is to show the admittance matrix for the specified sequence. One of the most important features of these matrix displays is the ability to right-click on the display to bring up additional display options in the [local menu](#). Perhaps one of the most important options on the local menu is the ability to export the Y-bus admittance matrix to a Matlab M file, which allows import of the matrix into Matlab for additional manipulation, such as inverting the matrix to get the equivalent sequence Z-bus matrix.



---

**Fault Analysis Bus Records**

---

This dialog has the same functionality available as [Case Information](#) displays. The purpose of this display is to tabulate the results of the fault analysis calculations. By default, the phase voltage magnitudes and angles are displayed. In addition, the sequence voltages and angles can also be added by modifying the display using the [Display/Column Options](#) dialog.

---

### **Fault Analysis Generator Records**

---

This dialog has the same functionality available as [Case Information](#) displays. The purpose of this display is to tabulate the results of the fault analysis calculations. By default, the phase current magnitudes are displayed for the terminal end of the generator. The phase current angles, as well as the sequence current magnitudes and angles, can be added by modifying the display using the [Display/Column Options](#) dialog. The magnitude and angle direction reference is always given as out of the generator and into the terminal bus.

---

**Fault Analysis Line Records**

---

This dialog has the same functionality available as [Case Information](#) displays. The purpose of this display is to tabulate the results of the fault analysis calculations. By default, the phase current magnitudes are displayed for each end of the branch. The phase current angles, as well as the sequence current magnitudes and angles, can be added by modifying the display using the [Display/Column Options](#) dialog. The magnitude and angle direction reference is always given as out of or away from a bus.

---

**Mutual Impedance Records**

---

The **Mutual Impedance Records** table is a [Case Information Display](#) and can be customized like any other case information display. The zero sequence mutual impedance records displayed in this table can be either read from a sequence data file, or created manually by choosing **Insert...** from the local menu.

The common fields displayed on the Mutual Impedance Records display are:

**L1 From Bus, L1 To Bus, and L1 Ckt ID**

These fields represent the from bus number, to bus number, and circuit identifier for the first mutually coupled line.

**L2 From Bus, L2 To Bus, and L2 Ckt ID**

These fields represent the from bus number, to bus number, and circuit identifier for the second mutually coupled line.

**Mutual R, Mutual X**

The mutual impedance, in terms of the resistance and reactance (per unit). The dot convention of the mutual impedance assumes the From bus of each line to be the dotted terminal, with the sign of the mutual impedance values being set according to this convention.

**L1 Mut. Start, L1 Mut. End**

The starting point and ending point of the mutually coupled portion of the first mutually coupled line. The values are between 0 and 1, and represent a position on the line as a percentage of the total line length. These fields are only used when evaluating an in-line fault to determine the affect of the mutual impedance on each side of the fault point on the line.

**L2 Mut. Start, L2 Mut. End**

The starting point and ending point of the mutually coupled portion of the second mutually coupled line. The values are between 0 and 1, and represent a position on the line as a percentage of the total line length. These fields are only used when evaluating an in-line fault to determine the affect of the mutual impedance on each side of the fault point on the line.

---

**Mutual Impedance Record Dialog**

---

The **Mutual Impedance Record** dialog can be used to modify or add zero sequence mutual impedance records to the sequence data for a case. When the dialog is opened using the **Show Dialog...** option from the [Mutual Impedance Records](#) table local menu, the information for the record selected in the table will automatically be displayed. The information for that record can be modified, or a different record can be selected by selecting different lines in the Line 1 and Line 2 Identifier sections. Note that the drop down list of buses for the From Bus fields always contain all the buses in the case. However, once the From Bus has been selected, the drop down list of the corresponding To Bus field will only contain bus numbers of buses that are connected to the From Bus. If a mutual impedance record already exists for the lines selected, the information for that record will be displayed. If a mutual impedance record does not exist for the selected lines, then the mutual impedance fields will display default values. When the default values are changed, and either Save or OK are selected, a new mutual impedance record is added to the data.

---

### **Fault Analysis Load Records**

---

This dialog has the same functionality available as [Case Information](#) displays. The purpose of this display is to tabulate the results of the fault analysis calculations. By default, the phase current magnitudes are displayed for the terminal end of the load. The phase current angles, as well as the sequence current magnitudes and angles, can be added by modifying the display using the [Display/Column Options](#) dialog. The magnitude and angle direction reference is always given as out of the bus and into the load.

---

**Fault Analysis Switched Shunt Records**

---

This dialog has the same functionality available as [Case Information](#) displays. The purpose of this display is to tabulate the results of the fault analysis calculations. By default, the phase current magnitudes are displayed for the terminal end of the switched shunt. The phase current angles, as well as the sequence current magnitudes and angles, can be added by modifying the display using the [Display/Column Options](#) dialog. The magnitude and angle direction reference is always given as out of the bus and into the switched shunt.

---

## Contouring

---

Simulator can create and animate a contour map of various system quantities, such as voltage magnitudes and angles, MW transactions, transmission loading, and real and reactive load. Such displays resemble a contour map of temperatures like you might see shown on a weather forecast. Contouring can significantly improve understanding of a large interconnected system, helping identify congestion pockets and Mvar-deficient regions and providing an overview of how power flows through the bulk power system.

The [Contour Options Dialog](#) controls Simulator's contouring capabilities. To access it, click the right mouse button on an empty area of the oneline and choose **Contouring** from the resulting [local menu](#), or choose **Contouring** from the [Active](#) ribbon group on the [Onelines](#) ribbon tab.



---

## Contouring Options

---

The Contour Options Dialog allows you to draw contour maps of many system quantities, such as bus voltages or angles, transmission line and interface MVA loadings, and transmission line and interface [PTDFs](#).

To access this dialog, click the right mouse button on an empty area of the oneline and choose **Contouring** from the resulting [local menu](#), or choose **Contouring > Contouring** from the [Active](#) ribbon group on the [Onelines](#) ribbon tab. The Contour Options Dialog has three tabs: the [Contour Type](#) Tab, the [Contour Type Options](#) Tab, and the [Custom Color Map](#) Tab.

---

## Contour Type

---

### Object

Simulator can contour several different values. To specify what Simulator should contour, first choose the type of display object; the options are Bus, Line, Interface, Area, Generator, Substation, Zone or Injection Group. Once the object is chosen, you must then choose the numerical value to be contoured by specifying it in the **Value** dropdown box.

Note: to contour a value for a type of object, representations of that type of object must be present on the oneline diagram. Choosing to contour an object type that is not represented on the diagram will result in no contour being drawn on the diagram.

### Value

Select the quantity to contour from the Value dropdown box or click the **Find...** button to find the desired field (it is normally easier to use the Find button). See NOTE at the end of this help topic.

### Filter Object Values

The name of the advanced filter that is currently applied to the contour. Click Define to create a new [Advanced Filter](#) to use during contouring. Applying a filter means that *only* objects that meet the filter will affect the contour image. Objects which do not meet the filter will be ignored.

### Data Points per Line

This specifies the number of data points used to represent the graphical line. The contouring algorithm is based on values at specific coordinates in the oneline space. When contouring transmission line or interface objects, the line is represented by a series of points spaced equally along the graphical line.

### No Data Color

This setting allows you to choose the contour color used for parts of the contour for which there are not objects or nor data for the type of contour selected. The choices for No Data Color are Specific Color, Color Map Percentage, and Background Color. By default, Specific Color is selected and set to white. If you wish to change the specific color to use, click on the color box to the right and choose a different color from the popup dialog. If you select Color Map Percentage, the Color Map % field will become enabled, and you can select a value from 0 to 100. The value you enter will associate the No Data Color with the color located at that percentage in the selected color map. Lastly, if Background Color is chosen, the No Data Color will always be whatever color has been set as the normal background color for the oneline diagram.

### Draw Color Key

Checking this box will cause the contour to draw a color key showing which colors are mapped to which values. You can also give the color key a title, unit label, and specify the number of digits to display in numerical values.

### Title

Title for the color key.

### Entry Labels

Units of the contoured value displayed on the color key.

### Dec. Pts.

Number of decimal places of the contoured value displayed on the color key.

### Scalar

Multiplication factor that can be applied to the values when drawing the color key. Normally this value should be 1.0.

### Use Equal Spacing For Discrete Maps

This option will draw the color key with equal spacing for all colors in the map, regardless of how close or distant the values the colors represent. This option only applies to discrete color maps.

### Color Map

Choose from various predefined color maps using the color map combo-box. A color map, along with the values specified, defines how values are mapped to a color on the contour image.

If a color map showing both high and low values is desired (such as for bus voltages), use of "Blue = Low, Red = High" is recommended. If a color map showing only high values is desired (such as for line flows), use of "Weather Radar, Nominal to High" is recommended.

A user may also define additional color maps by going to the [Custom Color Map](#) Tab.

### Reverse Color Map Colors

Check this check-box to reverse the colors of the selected color map, so the low color becomes the high color, and vice versa.

### Brightness

Modify the brightness track bar to change the brightness of the color map.

### Use absolute value

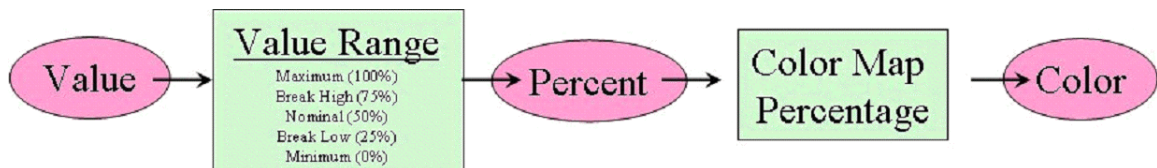
Check this check-box to use the absolute values of the quantity selected at the Value dropdown box (above).

### Ignore Above Max

Check this check-box to completely ignore values above the maximum percentage. This means that data which is larger than the Max % will not be used in calculating the contour image. This is similar to using an Advanced Filter to Filter Object Values, but provides a quick way to do so.

### Values

For all color maps that come with PowerWorld by default, the conversion of a value to a color is done in two steps. The value is first interpolated to a percentage by using up to 5 user-specified values (maximum, break high, nominal, break low and minimum) that correspond to specific percentages (100%, 75%, 50%, 25%, and 0%). This percentage is then interpolated through the color map and a color is determined.



User may define their own color maps which convert percentage to color, but they may also define color maps which directly translate values into colors without the intermediate calculation of a percent. For more information on this see the creation of a [Custom Color Map](#).

The value range values are described as follows:

**Maximum** The largest value allowed in the contour. All values above this will be mapped to the highest color. This value corresponds to 100% in the color map.

**Break High** This value is used by some color maps to highlight a lower limit. This value corresponds to 75% in the color map.

**Nominal** This value is the nominal value for the contour. Values around this will be mapped to the middle color. This value corresponds to 50% in the color map.

**Break Low** This value is used by some color maps to highlight a lower limit. This value corresponds to 25% in the color map.

**Minimum** The smallest value allowed in the contour. All values below this will be mapped to the lowest color. This value corresponds to 0% in the color map.

Note: a representation of the color map is shown to the right of the values.

### Ignore Below Min

Check this check-box to completely ignore values below the minimum percentage. This means that data which is smaller than the Min % will not be used in calculating the contour image. This is similar to using an Advanced Filter to Filter Object Values, but provides a quick way to do so.

### Ignore Zero Values

Check this box to completely ignore zero values in the contour. This is similar to using an Advanced Filter to Filter Object Values, but provides a quick way to do so. If there is a need to ignore values that are almost but not exactly zero, then create an Advanced Filter to Filter Object Values.

### Interpretation

This combo box specifies how to interpret the values of the data points. The options are:

**Fixed Values** The data point values are not modified. The maximum, minimum, nominal, and break values are the ones entered directly in the units of the value being contoured. This is what should normally be used.

**Dynamic Values** The data point values are not modified. However the maximum, minimum, nominal, and break values are determined dynamically from the data point values as follows: Maximum = Maximum data point value; Minimum = Minimum data point value; Nominal = Average data point value; Break High =  $(\text{Max} + \text{Average})/2$ ; and Break Low =  $(\text{Min} + \text{Average})/2$ .

**Standard Deviations** All the data point values will be used to determine a mean and standard deviation. The data point values will then be converted to represent the number of standard deviations they are from the mean. Thus a value equal to the mean will be changed to a 0, a value 1.5 standard deviations higher than the mean will be changed to 1.5, and so on.

**Percentiles** All the data point values will be sorted from lowest to highest. The value will then be set equal to the 100 times the sort location divided by the number of data points. Thus the highest value will be given a value of 100 and the lowest a value of 1.

### Save Contour Image with Oneline

Checking this box will allow a displayed contour to be saved with a oneline diagram. If a contour is saved with a oneline diagram, the next time the oneline diagram is opened the contour will automatically be redrawn as well. The displayed contour is saved as a Bitmap image.

### Continuously Update Contours

Normally contouring is only done on a snap shot of the power system state. However, you can also set PowerWorld to automatically update the contour every time the display is redrawn. In this way, an animation of the contour can be created. If you would like to create this animation, simply check the Continuously Update Contours checkbox. Note, however, that this will slow down the animation of the display, as the program must recalculate the contour at each step. If this slows down your display too much, try lowering the contour resolution to speed it up.

**Note Regarding Values**

Contours of most values create an image where the color around a data object is primarily related to the value of only that object. Some values however create "density-like" contours, where the color is related to the sum of the data object's values nearby. These include:

- Bus / Load MW
- Bus / Load Mvar
- Bus / Load MVA
- Bus / Cust Expr (Density)
- Area / Pos Spin Reserve
- Area / Neg Spin Reserve

---

## Contour Type Options

---

### Object

Simulator can contour several different values. To specify what Simulator should contour, first choose the type of display object; the options are Bus, Line, Interface, Area, Generator, Substation, Zone or Injection Group. Once the object is chosen, you must then choose the numerical value to be contoured by specifying it in the **Value** dropdown box.

Note: to contour a value for a type of object, representations of that type of object must be present on the oneline diagram. Choosing to contour an object type that is not represented on the diagram will result in no contour being drawn on the diagram.

### Value

Select the quantity to contour from the Value dropdown box or click the **Find...** button to find the desired field (it is normally easier to use the Find button). Note: see [Contour Type](#) for more information.

### Filter Object Values

The name of the advanced filter that is currently applied to the contour. Click Define to create a new [Advanced Filter](#) to use during contouring. Applying a filter means that *only* objects that meet the filter will affect the contour image. Objects which do not meet the filter will be ignored.

### Influence Region

This track bar determines how far away each data point influences the contour image. A larger influence region results in each data point influencing more of the contour at the expense of longer screen refresh times.

### Use Dynamic Influence Region

Dynamic influence distance determines how far out the contour should go when determining which buses influence the contour value for a screen point. The actual distance is the minimum of either 1) a common value for all screen points that depends upon user parameters [e.g., the dynamic region points value]), and now 2) the distance that includes the number of buses associated with the dynamic influence field. The primary reason for this option is speed, particularly when zoomed in on dense portions of the display. You should not see much impact on the contour itself.

### Kind of Value

This option allows you to choose to contour based on the Actual Value or the Density Value. The Actual Value uses the weighted average of the data for computing the contour. The Actual Value method is most commonly used for contouring in Simulator. On occasion, the Density of Values method does not work as well. One example is contouring generator MW values. If you have four buses in close proximity, each with 100 MW of generation, and compare the contour with a single bus with 400 MW of generation, the contour based on the weighted average will look drastically different, despite the amount of generation being the same in each region. Using the Density Value option will correct the disparity, and the contour around these two different groups of generation would look basically the same.

### Use Fade to Value

Checking this check box will allow to use the Fade to Value and Begin Fade Percentage options.

### Fade to Value

The value to which a data point's value fades as it moves away from its location.

**Begin Fade Percentage**

While moving away from a data point, the data point's value decays towards the "Fade to" Value. The Begin Fade Percentage specifies when the contour starts to fade as a percentage of the largest distance for which this data point influences the contour.

**Contour Resolution**

This value determines the relative resolution of the contour. Increasing the contour resolution increases the level of detail represented on the map but will lengthen screen refresh times. Reducing the screen refresh time will yield less detail and shorter screen refresh times.

**Low Resolution Display when Zooming or Panning****Do Low Resolution Speedup**

Check this option to enable drawing contours in low resolution while zooming or panning. Low resolution images are drawn much faster than high resolution images.

**Low Resolution Normalized Percent**

Set a percentage of resolution between 0.01 (1%) and 1 (100%). A smaller value translates to a lower resolution image.

**Time Delay in Seconds before Redrawing at Full Resolution**

Set the number of seconds Simulator will pause before returning the contour to full resolution following zooming and panning.

**Minimum Resolution in Pixels for Low Resolution Displays**

Specify the minimum resolution allowed (in pixels) when Simulator draws the low resolution image.

**When Animating, Only Update Contour if Something Changed**

As it describes, checking this option will suppress redrawing the contour during animation unless one or more of the contoured values changes.

---

## Custom Color Map

---

### Color Map

Choose from various predefined color maps using the color map combo-box. A color map, along with the values specified, defines how values are mapped to a color on the contour image.

If a color map showing both high and low values is desired (such as for bus voltages), use of "Blue = Low, Red = High" is recommended. If a color map showing only high values is desired (such as for line flows), use of "Weather Radar, Nominal to High" is recommended.

A user may also define additional color maps by going to the Custom Color Map Tab.

### Reverse Color Map Colors

This check-box reverse the mapping of the color map, converting the colors corresponding to the high values into the colors for the low values, and vice-versa.

### Brightness

Modify the brightness track bar to change the brightness of the color map.

### Make Discrete Color Map

Check this check-box to make a discrete color map, that is without having smooth transitions between colors.

### Contour Type Values to Use

These check-boxes signify which values from the Contour Type Values Tab are used by the Color Map. There must be at least two contour type values checked.

### Color Grid

The color grid on the right side of this page allows you to change the colors for each percentage breakpoint. In addition, you can add or delete breakpoints as well.

To change the color for a specific breakpoint, simply left-click on the color for the breakpoint you want to change. The Color dialog will open, and you can choose a new color for that breakpoint.

To add a breakpoint, right-click on a breakpoint position above or below where you would like the new breakpoint inserted. A popup menu will open, and you can select Add Above or Add Below, depending on where you wish the new breakpoint to be. Simulator will insert the breakpoint, and will automatically set the color and percentage at the midpoint between the two breakpoints above and below the inserted breakpoint. You can then click on the color to change it, or click on the percentage to type in a new value. The new percentage value should be between the values of the adjacent breakpoints.

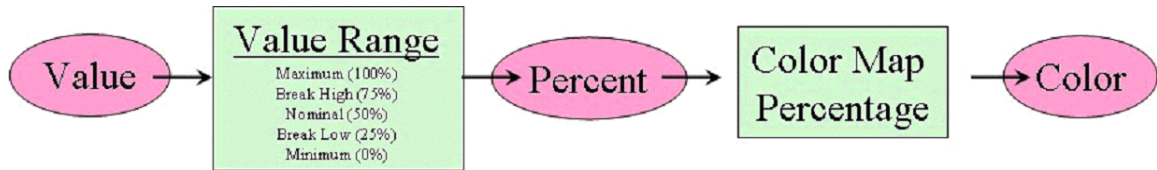
To delete a breakpoint, right-click on the breakpoint you wish to delete, and choose Delete from the popup menu.

### Color By...

This option allows the user to interpret the breakpoints values as percentage values or as direct values.

For all color maps that come with PowerWorld by default, the conversion of a value to a color is done in two steps. The value is first interpolated to a percentage by using up to 5 user-specified values (maximum, break high, nominal, break low and minimum) that correspond to specific percentages (100%, 75%, 50%, 25%, and 0%). This percentage is then interpolated through the color map and a color is determined.





Color maps which directly translate values into colors without the intermediate calculation of a percent may also be created. This is done by setting Color By to *Value*.

#### **Save As...**

To save the present color map as a new color map, click this button. Then specify a name for the new color map.

#### **Save**

To save changes that have been made to the present color map, click this button.

#### **Rename**

To rename the present color map, click this button.

#### **Delete**

To delete the present color map, click this button.

#### **Store Color Maps in File**

To store all custom color maps in a file for loading into another case, click this button. If you have saved any custom color maps with the current case, you will be prompted to choose a file name and location for saving the custom color maps. Color Maps are saved in the [Auxiliary File Format](#).

#### **Load Color Maps from File**

If you have created custom color maps in a different case and saved them to a file, you can click this button to load those color maps into your current case. Color Maps are loaded in the [Auxiliary File Format](#).

---

## Functional Description of Contour Options

---

### Functional Description of the Contour Options

The previous help topics discussed basic contour options in the order they are arranged on the dialog. The dialog is arranged so that the most important options are on the first tab and other on the Contour Type Options tab. This help topic discusses the options in a manner which better describes how the contour is actually created. A contour is calculated generally by a five-step process

1. Build a list of all possible data points (graphical locations)
2. Remove items that meet criteria from the list of data points
3. Assign a value to each data point
4. Calculate "virtual values" on a grid of points
5. Convert each "virtual values" into a color to create the contour image

#### Step 1: Build a list of data points (graphical locations)

The following options determine a list of potential data points (could also be called graphical locations) which will be used to calculate the contour image.

**Object** - Simulator can contour several different values. To specify what Simulator should contour, first choose the type of object. This corresponds to the type of display object that is drawn on the oneline diagram. For instance if you want to contour a substation value, then you must have substations drawn on your diagram. This selection narrows the set of quantities that can be contoured, which is specified in the Value dropdown box.

**Data points Per Line** - The contouring algorithm for lines is no different than for points, except that each line is represented by several points. For objects which are represented by graphical lines, this option will specify the number of data point which should be used to represent the line.

#### Step 2: Remove items that meet criteria from the list of data points

After a complete list of potential data points is made, there are then several options for filtering out things from this list which you do not want to effect the calculation of the contour image.

**Filter** - If you wish for the contour to be limited to only certain devices that meet specific criteria, click on this button to define an Advanced Filter for the contour

**Ignore Above Max** - Check this check-box to completely ignore values above the maximum percentage. This means that data which is larger than the Max % will not be used in calculating the contour image

**Ignore Above Min** - Check this check-box to completely ignore values below the minimum percentage. This means that data which is smaller than the Min % will not be used in calculating the contour image

**Ignore Zero Values** - Check this check-box to completely ignore values that are zero.

### Step 3: Assign a value to each data point

After Step 2, a list of potential data points has been created and a value must now be assigned to each data point. The following options specify how this is done.

**Value** - select the quantity to contour from the Value dropdown box or click the Find Value button to find the desired field.

**Use absolute value** - This check-box will modify the Value specified so that it uses the absolute value.

**Interpretation** – When interpretation is set to either Standard Deviations or Percentiles, then the value of the data point will be modified.

If **Standard Deviations** is chosen, then all the data point values will be used to determine a mean and standard deviation. The data point values will then be converted to represent the number of standard deviations they are from the mean. Thus a value equal to the mean will be changed to a 0, a value 1.5 standard deviations higher than the mean will be changed to 1.5, and so on. Similar,

If **Percentiles** is chosen, then all the data point values will be sorted from lowest to highest. The value will then be set equal to the 100 times the sort location divided by the number of data points. Thus the highest value will be given a value of 100 and the lowest a value of 1.

### Step 4. Calculate "virtual values" on a grid of points

After Step 3, we now have a list of data points and a value assigned to each data point. We now must create a grid of points that represents "virtual values" throughout the entire graphical space. The values on this grid will be calculated based on the data point values

**Contour Resolution** - setting determine the size of the grid of points which will be superimposed on the present oneline diagram. The higher the resolution the more number of grid points will be used to represent the contour (and thus make the calculation of the contour image slower)

**Influence Region** - each data point value will effect only the grid locations that are "near" it. The distance that is considered "near" is determined by the Influence Region. Setting the

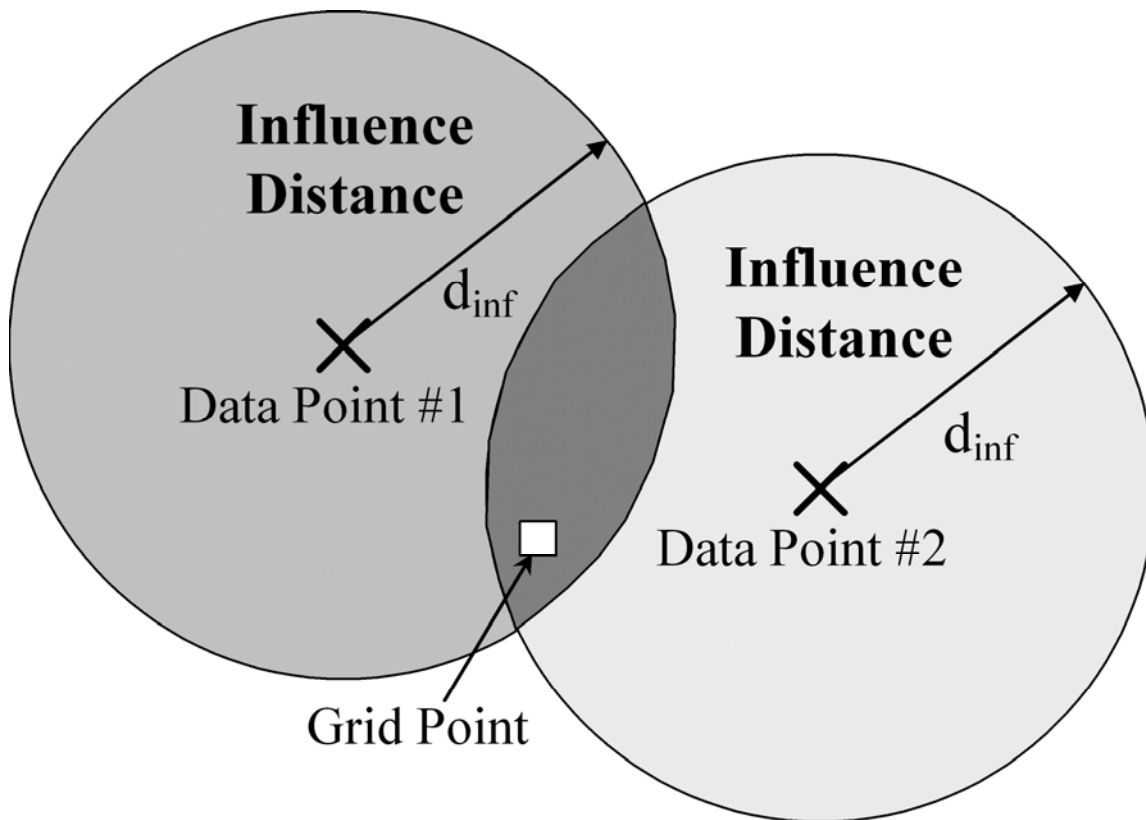
influence region higher will result in each data point effecting a larger portion of the contour image (and thus making the calculation of the contour image slower)

**Kind of Value** - The calculation of the virtual value at a particular grid point is done by first building a list of data points that are within the "influence distance" of the grid point. The Kind of Value setting determines what process is used to calculate the virtual value from this list of values.

**Actual Value (Weighted Average)** - The virtual value is calculated as the weighted average value, weighted by the distance from the grid point. This means that virtual value half-way between two data points will be the average of the two data point values.

**Density of Values (Weighted Sum)** - The virtual value is calculated as the sum of value that within the influence region. This means that virtual value half-way between two data points will be the sum of the two data point values.

**Actual Value (Only Closest)** - The virtual value is assigned as the value of the closest data point.



**Fade to Value and Begin Fade Percentage** - In the middle of a contour image the colors look consistent because the virtual values are calculated using a good number of data points which surround it. Sometimes at the edge of a contour image however, the colors can become skewed because there are very few data points influencing it. Using a fade to value can help this situation, but will skew the entire contour image in general instead. Without

using the fade to value, the raw data point values will be used when calculating virtual value.

When using the fade to value, while moving away from a data point, the data point's value decays towards the "Fade to" Value. The Begin Fade Percentage specifies when the contour starts to fade as a percentage of the largest distance for which this data point influences the contour.

### Step 5. Convert each "virtual values" into a color to create the contour image

After Step 4, a grid of virtual values has been calculated. At this point, must specify how these numbers map to colors. This done through the user of a Color Map.

**Color Map** - Choose from various predefined color maps using the color map combo-box. A color map, along with the values specified, defines how values are mapped to a color on the contour image. A user may also define additional color maps by going to the Custom Color Map Tab.

**Brightness** - this value is used to brighten or darken the colors specified in the color map.

**Values** - These values along with the color map define how to convert your values into a color for the contour. The values are:

**Maximum** - The largest value allowed in the contour. All values above this will be mapped to the highest color. This value corresponds to 100% in the color map.

**Break High** - This value is used by some color maps to highlight a lower limit. This value corresponds to 75% in the color map.

**Nominal** - This value is the nominal value for the contour. Values around this will be mapped to the middle color. This value corresponds to 50% in the color map.

**Break Low** - This value is used by some color maps to highlight a lower limit. This value corresponds to 25% in the color map.

**Minimum** - The smallest value allowed in the contour. All values below this will be mapped to the lowest color. This value corresponds to 0% in the color map.

**Interpretation** - Most frequently this option will be set to **Fixed Values** meaning values for maximum, break high, etc... are entered directly in the units of the value being contoured. If Interpretation is set to **Standard Deviations** or **Percentiles** then as mentioned earlier the values mean something different. The options **Dynamic Values** will process this list of data point values and automatically set the values as follows: Maximum = Maximum data point value; Minimum = Minimum data point value; Nominal = Average data point value; Break High =  $(\text{Max} + \text{Average})/2$ ; and Break Low =  $(\text{Min} + \text{Average})/2$ .

**No Data Color** - It is likely that some the grid points will not be within the influence of any of the data points. The color of these points is determined by no data color setting.

---

## Power Transfer Distribution Factors

---

The Power Transfer Distribution Factor (PTDF) display is used to calculate the incremental distribution factors associated with power transfers between two different areas or zones. These values provide a linearized approximation of how the flow on the transmission lines and interfaces change in response to transaction between the Seller and the Buyer. These values can then be visualized on the onelines using animated flows (see below for details). The transaction for which the PTDFs are calculated is modeled by scaling the output of all generators on [AGC](#) in the source and sink areas in proportion to their relative [participation factors](#). Generators in the source area increase their output, while generators in the sink area decrease their output.

An important aspect to consider in calculating the PTDF is how the losses associated with the transfer are allocated. Simulator assumes that the Seller increases the output of its generators by 100% of the transfer amount, while the Buyer decreases the output of its generators by 100% **minus any change in system losses**. In other words, the Buyer accounts for the entire change in the system losses. Of course it is possible that a transfer may result in decreased system losses; for that case, the Buyer's generation will be greater than 100% of the transfer.

### To Calculate the Power Transfer Distribution Factors:

- Perform an initial Power Flow Solution.
- In Run Mode, click on the Tools ribbon tab and select **Power Transfer Distribution Factors (PTDFs)** from the **Sensitivities** menu on the **Run Mode** ribbon group to open the [Power Transfer Distribution Factors Dialog](#).
- Supply the requested information on the [Power Transfer Distribution Factors Dialog](#) and click the **Calculate PTDFs** button. The distribution factors are calculated and displayed for the element set of your choice in the table at the bottom of the dialog.

The animated flows that appear on the oneline diagram may represent either actual flows or PTDF values. To specify that the display should show distribution factors, click the button labeled **Visualize PTDFs**. Once this button is clicked, the flow arrows on all open onelines will represent distribution factors, and the caption of the button will change to **Visualize Actual Power Flows**. Click the button again to visualize actual power flows instead of distribution factors.

Note that when calculating PTDF values for interfaces that include contingent elements, the PTDF values reported are actually what are referred to as an Outage Transfer Distribution Factor (OTDF). See [Line Outage Distribution Factors \(LODFs\)](#) for more information.

---

## Power Transfer Distribution Factors Dialog

---

The PTDF Dialog enables you to control and to view the results of [power transfer distribution factor calculations](#). To access this dialog go to the [Tools](#) ribbon tab, and choose **Sensitivities > Power Transfer Distribution Factors** from the [Run Mode](#) ribbon group. This is only available in [Run Mode](#).

The dialog has the following options:

### Linear Calculation Method

PTDFs may be calculated using either the full power flow Jacobian or only a portion of it. If you select the **Linearized AC Approximation** option, the sensitivity of the monitored element's flow will be calculated as a function of both its real and reactive power components to the voltage magnitude and angle of its terminal buses. When using the AC method, losses are included in the calculation. Simulator assumes that the change in losses is taken care of by the Buyer.

If you instead select the **Lossless DC Approximation** option, branch flow sensitivity is calculated by estimating the real power that flows through the monitored element only from the difference in angles measured across its terminals. This method assumes that there are no losses.

The **Lossless DC with Phase Shifters Approximation** option is similar to the **Lossless DC**, except additional constraints are placed on the calculations that assume that the change in flow across active phase-shifting transformers is zero.

Note: The Lossless DC methods will be affected by the **DC Power Flow Model** settings in [Power Flow Solution: DC Options](#). Click on the **DC Model Options** button to open the dialog which sets these options.

### Directions

This option allows you to choose to define a single direction PTDF using the **Seller** and **Buyer Type** related fields, or to define multiple transfer directions between many different entities. If you choose to use multiple directions, the **Seller** and **Buyer Type** fields are replaced by the [Direction Records display](#) for viewing and defining directions.

### For Single Direction: Seller Type, Buyer Type

Distribution factors can be calculated for power transfers between combinations of areas, zones, super areas, participation groups, or to a slack bus. Use the *seller type* and *buyer type* options to indicate the type of the selling and purchasing entities. These fields are only present for single direction PTDFs. When choosing Area, Zone, or Super Area, this really means all generators in the Area, Zone or Super Area weighted by the generator participation factor.

### For Single Direction: Seller, Buyer

These dropdown boxes allow you to select the selling and buying entities. Their contents are filled when you select the seller and buyer types. These fields are only present for single direction PTDFs.

### For Single Direction: Reverse Buyer/Seller

Click this button to re-calculate PTDFs for the direction that is the reverse of the direction currently shown. For example, if you have just calculated PTDFs for a transaction from area A to area B, press this button to calculate and display PTDFs for a transaction from area B to area A. This option is only present for single direction PTDFs.

### Calculate PTDFs

Click on this button to update the PTDF values. The results table will reveal the latest calculations.

### Automatically Recalculate

If checked, the PTDFs are automatically recalculated every time the power flow is solved.

### Calculate MW-Distance

Click this button to open the **MW \* Distance Calculations** form. This form allows you to calculate MW \* Distance values for the transaction for which you calculated PTDFs. See [MW-Distance Calculations](#) for more information.

### Increase in Losses

This is a read-only field that indicates the change in system losses caused by the transfer from the selling area to the buying area. The change is expressed as a percentage of the transfer amount. This will only be non-zero when using the **Linearized AC** calculation method.

### Use Area/Zone Filters

If this box is checked, then the results table at the bottom of the dialog will include only records associated with devices located in areas or zones included in the [area/zone/owner filter](#) set.

### Only Show Above %

Restricts the result set to show only those PTDFs that exceed a specified value.

### Visualize Actual Power Flows, Visualize PTDFs

Select to toggle the onelines between showing the actual power flows and the PTDF flows. Selecting this button changes the Flow Visualization field for all the visible onelines. You can also change this field manually using the [Online Display Options](#) Dialog.

### Tables of Results

The tables of results occupy the bottom of the PTDF Dialog. They are a set of [case-information displays](#) and thus share many characteristics common to all other case information displays. The tables will show results for lines/transformers, interfaces, areas, zones, generators, and phase shifters. The tables feature a [local menu](#) from which you can print, copy, or modify its records as well as view the [information dialog](#) of its associated element. You can also sort the area records by clicking on the heading of the field by which you want to sort.

### Lines/Transformers

Shows the transaction distribution factors for the lines and transformers. The following fields are shown:

#### From Bus #, From Bus Name, To Bus #, To Bus Name, Circuit

Identifiers for the transmission line or transformer.

#### From % PTDF

Distribution factor associated with the MW flow at the "from bus" end of the line or transformer, specified as a percentage of the transaction amount.

#### To % PTDF

Distribution factor associated with the MW flow at the "to bus" end of the line or transformer, specified as a percentage of the transaction amount.

#### % Losses



Shows the percentage of the PTDF assigned as losses.

**Nom KV (Max) and Nom KV (Min)**

Displays the maximum and minimum nominal voltages for the line. This is useful for identifying transformers and which end of the PTDF relates to which nominal voltage.

**Interfaces**

Shows the transaction distribution factors for the [interface records](#). The following fields are shown:

**Interface Name**

Alphanumeric identifier for the interface.

**Interface Number**

Numeric identifier for the interface.

**% PTDF**

Distribution factor associated with the MW flow through the interface, specified as a percentage of the transaction amount. A positive value indicates the transaction would result in an increase in the flow through the interface.

**Interface MW Flow**

Amount of real power flowing on the interface.

**Has Contingency**

Signifies if an element of the interface is violating a limit.

**Areas and Zones**

Shows the impact the transaction has on the losses for the area or zone. The following fields are shown:

**Area/Zone Number and Name**

Number and name identifiers for the area or the zone

**Losses %**

Change in the losses in the area or zone, specified as a percentage of the transaction amount. A **positive number** indicates that the transaction would result in **increased** losses in the area or zone, while a **negative number** indicates that the transaction would result in **decreased** losses.

**Gen Chg %**

Total change in all of the generators in area or zone, specified as a percentage of the transaction amount. For areas, this field should show 100% in the selling area, and 100% minus the change in system losses in the buying area.

**Generators**

Shows the marginal participation of each generator in the transaction. The following fields are shown:

**Bus Number, Bus Name, Gen ID**

Generator's terminal bus number and alphanumeric identifier, and the id for the generator.

**Area Number, Area Name**

Name and number of the generator's area.

**Gen Change %**

Assumed participation of the generator in the transaction, specified as a percentage of the transaction amount. This value is directly proportional to the participation factor for the generator, provided the generator is available for AGC and is free to move in the specified direction (i.e., is not at a MW limit). The generator's participation factor and AGC status are modified on the [Generator Dialog](#), which can be displayed by right-clicking anywhere in the record's row in the table and selecting the *Show Dialog* option.

### **Phase Shifters**

Shows the transaction distribution factors for the phase shifters. This applies when using the Lossless DC with Phase Shifters calculation method. The following fields are shown:

#### **From Number, From Name, To Number, To Name, Circuit**

Identifiers for the phase shifter.

#### **Status**

Indicates whether or not the phase shifter is in-service.

#### **Phase (Deg)**

The current phase angle of the transformer.

#### **XF Auto**

Indicates if the phase shifter is currently enabled for automatic control.

#### **Deg per MW**

This field indicates the amount of angle change, in degrees, that would be required to keep the flow across the transformer constant for a one MW transfer between the seller and the buyer.

#### **Tap Min, Tap Max**

The minimum and maximum tap positions for the phase shifter.

---

## Line Outage Distribution Factors (LODFs)

---

Line Outage Distribution Factors (LODFs) are a sensitivity measure of how a change in a line's status affects the flows on other lines in the system. On an energized line, the LODF calculation determines the percentage of the present line flow that will be show up on other transmission lines after the outage of the line. For example, consider an energized line, called LineX, whose present MW flow is 100 MW. If the LODFs are found to be

LODFs for LineX outage

LineX -100%

LineY + 10%

LineZ - 30%

This means that after the outage of LineX, the flow on LineX will decrease by 100 MW (of course), LineY will increase by 10 MW, and LineZ will decrease by 30 MW. The "flow on Line X" here means the flow at the *from* bus going toward the *to* bus.**Error! Hyperlink reference not valid.**

Similarly, sensitivities can be calculated for the insertion of a presently open line. In this case, the LODF determines the percentage of the post-insertion line flow that will come from the other transmission line after the insertion. The "LODF" might better be named a Line Closure Distribution Factor (LCDF) in this case.

### To calculate the LODFs:

- Perform an initial [Power Flow Solution](#).
- In [Run Mode](#), select **Sensitivities > Line Outage Distribution Factors (LODFs)** from the **Rune Mode** ribbon group on the [Tools](#) ribbon tab to open the [Line Outage Distribution Factors Dialog](#).
- Supply the requested information on the [Line Outage Distribution Factors Dialog](#) and click the **Calculate LODFs** button.

### What else are LODFs used for?

LODFs are used extensively when modeling the linear impact of contingencies in Simulator. This is true for the calculation of [PTDFs](#) for interfaces which contain a contingent element, as well as when performing Linear [ATC analysis](#) that includes branch contingencies.

When calculating "PTDF" values for interfaces that include [contingent elements](#), the PTDF values reported are actually what are referred to as an Outage Transfer Distribution Factor (OTDF). An OTDF is similar to PTDF, except an OTDF provides a linearized approximation of the *post-outage* change in flow on a transmission line in response to a transaction between the Seller and the Buyer. The OTDF value is a function of PTDF values and LODF values. For a single line outage, the OTDF value for line x during the outage of line y is

$$OTDF_x = PTDF_x + LODF_{x,y} * PTDF_y$$

where  $PTDF_x$  and  $PTDF_y$  are the PTDFs for line x and y respectively, and  $LODF_{x,y}$  is the LODF for line x during the outage of line y. More complex equations are involved when studying contingencies that include multiple line outages, but the basic idea is the same.

When performing Linear [ATC analysis](#) along with calculating OTDFs, Simulator determines the linearized approximation of the post-outage flow on the line. This is similarly determined as

$$OutageFlowX = PreOutageFlowX + LODF_{x,y} * PreOutageFlowY$$

where  $\text{PreOutageFlowX}$  and  $\text{PreOutageFlowY}$  are the pre-outage flow on lines  $x$  and  $y$ .

---

## Line Outage Distribution Factors Dialog

---

The LODF Dialog enables you to control and to view the results of Line Outage Distribution Factor calculations. You access this dialog by selecting **Sensitivities > Line Outage Distribution Factors (LODFs)** from the **Run Mode** ribbon group on the [Tools](#) ribbon tab in [Run Mode](#) only.

The dialog has the following options:

### Output Option

Choose Single LODF to calculate only the sensitivities related to the status change of a single transmission line. Choose LODF matrix to calculate a matrix of sensitivities.

### For Single LODF: Specify Near Bus, Far Bus

Specify the line whose status modification you would like to determine sensitivities to. The direction of the near/far chosen is important. Switching the near and far bus around will change the sign of all the LODF sensitivities.

### For Single LODF: Action

Check Outage Sensitivities to determine sensitivities for the outage of a line.

Check Closure Sensitivities to determine sensitivities for the closure of a presently outaged line. The LODF might better be named a Line Closure Distribution Factor (LCDF) in this case.

### For LODF Matrix: Specify Lines to Process (Outage or Closure) and Lines to Monitor

Choose which lines to process and which lines to monitor by choosing either all AC lines, only those that meet the area/zone/owner filters, only lines with Selected field set to YES, or only those lines that meet an [Advanced Filter](#). After choosing **Calculate LODFs**, each row of the results represents the line being outaged/closed. Each column of the results represent the line begin monitored and the corresponding LODF/LCDF for that line.

### Linear Calculation Method

#### Linearized AC

This calculation method is not available for LODF sensitivities.

#### Lossless DC

Uses the DC power flow approximation.

#### Lossless DC with Phase Shifters

Check this to include the impact of phase shifter controllers with the Lossless DC calculation. By checking this, it is assumed that operating phase shifters will maintain their control requirements after the line outage.

Note: The Lossless DC methods will be affected by the **DC Power Flow Model** settings in [Power Flow Solution: DC Options](#). Click on the **DC Model Options** button to open the dialog which sets these options.

### Calculate LODFs

Click this to calculate the LODFs and update the display.

### Advanced LODF Calculation

Opens the [Advanced LODF Calculation](#) dialog for setting additional options for the calculation.

### LODFs Tab

After calculating a LODF Matrix, each row of the results represents the line being outaged/closed. Each column of the results represent the line begin monitored and the corresponding LODF/LCDF for that line.

When calculating LODFs for a single branch, This tab contains a table showing a list of the lines in the case. Since this table us another variety of the [Case Information Displays](#), you may interact with it in a familiar manner. Click on any of the field headings to sort by that field. Right-click on the display to call up the display's [local menu](#). From the [local menu](#), you can print the violations, copy the violation records to the Windows clipboard for use with another application, modify the format and content of the listing, view the information dialog of the respective element, and view the [Quick Power Flow List](#) or [Bus View Display](#).

The default fields for the tab are as follows

#### From Bus Number and Name

"From" bus number and name. Right- clicking on either of these fields allows you to see all the flows measured at the "from" bus using the [Quick Power Flow List](#) or [Bus View Display](#) local menu options.

#### To Bus Number and Name

"To" bus number and name. Right-clicking on either of these fields allows you to see all the flows into the "to" bus using the [Quick Power Flow List](#) or [Bus View Display](#) local menu options.

#### Circuit

Two-character identifier used to distinguish between multiple lines joining the same two buses.

#### %LODF

The LODF value for the line.

#### From MW, To MW

The present MW flows on the line at the "from" bus and the "to" bus.

#### From CTG MW, To CTG MW

The projected MW flows after the change in line status on the line at the "from" bus and the "to" bus.

Note: LODFs are always calculated using a DC power flow technique.

---

## Advanced LODF Calculation Dialog

---

The Advanced LODF Calculation Dialog can be opened by clicking on the Advanced LODF Calculation button found on the [Line Outage Distribution Factors \(LODFs\) dialog](#). The advanced calculation dialog allows you to calculate the [LODFs](#) for several different contingent lines in one batch process. First, you need to use the [Contingency Analysis Tool](#) to define which contingent lines to use (all contingencies that contain a single branch outage will be used.) Also, only lines that are being monitored will have their LODFs calculated. See [Limit Monitoring Settings](#) to change which lines are monitored.

The output from the advanced LODF calculation process is saved to a file for import into another program for analysis. This dialog defines how to save the Advanced LODF results to a file for importing in another program.

### Format to Save in

You can choose to save the advanced LODF results as "Monitored Branch, Contingency" pairs for PROMOD, or as a matrix in a comma-delimited text file. The comma-delimited text file is useful for loading the results into a spreadsheet program.

### Only save pairs with an LODF whose absolute value is greater than

This field allows you to filter out elements whose LODF is below a certain value.

### Only Include Monitored Branches whose MW flow increases

This field allows you to filter out monitored elements whose MW flow did not increase in the LODF calculation.

### Maximum Columns Per Text File

This field is important when you are saving a matrix in a comma-delimited text file. Most spreadsheet programs have limits on the number of columns they can display. The default is 256, which happens to be the maximum number of columns allowed in Excel.

### LODF Number Format

You can choose to have the LODF values stored in either scientific notation, or as a specified length decimal number.

### File Name

You must enter a file name and location or **Browse** for a file for saving the LODF data.

---

**Directions Display**

---

The Directions Display appears in the upper-right corner of the PTDF window when you select Multiple Directions for the PTDF Type. The Directions Display is a [case information display](#) that allows directions to be defined for performing multiple direction [Power Transfer Distribution Factors](#). The Directions Display allows you to insert, delete, and modify directions using options available from the display's [local menu](#). When directions are modified or inserted individually, the Directions Dialog will be displayed for entering the information. In addition to individually defining directions, they can also be automatically inserted using the Auto Insert Directions dialog.

Each record in the display shows the following default information:

**Number, Name**

A unique number and name given to the defined direction.

**Source Type, Source Name**

The name and type of the direction source.

**Sink Type, Sink Name**

The name and type of the direction sink.

**Include**

Determines if the direction is to be analyzed when calculating the multiple direction PTDFs.

**Processed**

Displays if the direction has already been processed for a multiple direction PTDF analysis.



---

## Directions Dialog

---

Directions are objects that are defined and used when computing a [Power Transfer Distribution Factor](#) using multiple directions.

The **Directions Dialog** can be used to insert a new direction or to modify the information for an existing direction. This dialog can be called by choosing **Insert** or **Show Dialog** from the [Directions Display](#) local menu.

The options that can be set from this dialog include:

### **Name of Direction**

The name for the direction. If you are entering a new direction, you can enter a new direction name. If you are modifying an existing direction, the name of the currently viewed direction will be displayed. A drop down list shows a list of currently defined direction names, and choosing one from the list will display that direction's information in the dialog. The up and down arrows next to the field also allow you to scroll through the list of defined directions.

### **Direction Number**

If you are entering a new direction, you can put in a new direction number. If you are viewing or modifying an existing direction, the direction's number will be displayed.

### **Source**

Select the type of the transfer direction source. Once the type has been selected, the [advanced find](#) list will be enabled, allowing you to search for the specific object or group of objects to set as the direction source.

### **Sink**

Select the type of the transfer direction sink. Once the type has been selected, the [advanced find](#) list will be enabled, allowing you to search for the specific object or group of objects to set as the direction sink.

### **Include in list of monitored directions**

Determines if the direction is to be analyzed when calculating the multiple direction PTDF's.

---

## Auto Insert Directions

---

Multiple directions can be automatically inserted for PTDF studies using the **Auto Insert Directions** dialog. This option can be selected from the local menu of the [Directions Display](#) by right-clicking and choosing **Direction Records > Auto Insert Directions**.

The layout of the dialog is as follows:

### Type of Direction

There are six types of directions that can be automatically defined. **Area to Slack**, **Zone to Slack** and **Injection Group to Slack** will define directions from areas, zones or injection groups to the slack bus of the system. **Area to Area**, **Zone to Zone** and **Inj. Group to Inj. Group** will define directions from areas to other areas, zones to other zones, or injection groups to other injection groups.

### Delete Existing Directions

When checked, any previously defined directions will be deleted before the new directions are automatically inserted. If not checked, then automatically inserted directions will be added to the list of previously defined directions. By not deleting existing directions before automatically inserting new directions, it is possible to have more than one direction defined with the same source and sink.

### Only Insert for Areas or Zones with Display Filters Set

If checked, then only Areas and Zones with their [Area/Zone/Owner Filters](#) set to **Yes** will be used when automatically inserting directions.

### Starting Number, Increment By

The **Starting Number** will be the first number used when automatically numbering the automatically inserted directions. Each subsequent direction added will be numbered according to the **Starting Number** and the **Increment By** value.

### Insert Directions

**Insert Directions** will perform the automatic insertion routine for the directions, according to the defined options.

---

## Flow and Voltage Sensitivities

---

To access the Flows and Voltages Sensitivities dialog go to the [Tools](#) ribbon tab, and choose **Sensitivities > Flows and Voltages Sensitivities** from the [Run Mode](#) ribbon group. This is only available in [Run Mode](#).

### Single Meter, Multiple Transfers

The Single Meter, Multiple Transfers Tab of the Flows and Voltages Sensitivities Dialog shows the effect an additional injection of real or reactive power at a bus has on real, reactive, or complex power flow on a particular line or interface, or on the voltage of a selected bus. The grid that occupies the bottom of the dialog lists each bus in the system, subject to the [Area/Zone/Owner Filter](#) settings. This grid is a [case information display](#) and thus shares properties and controls common to all other case information displays. The *P Sensitivity* field indicates the effect a 1 MW increase in real power at the bus has on the flow (either MW, Mvar, or MVA flow, as dictated by the **Flow Type** setting) or voltage on the device identified by the **Device Identifier**. Likewise, the *Q Sensitivity* field indicates the effect a 1 Mvar increase in reactive power at the bus has on the flow (either MW, Mvar, or MVA flow, as dictated by the **Flow Type** setting) or voltage (in per unit) on the device identified by the device identifier. All injections assume that the power is absorbed at the island slack bus.

Use the **Device Type** control to indicate whether the sensitivities are to be calculated for a line/transformer, an interface, or a bus. Use the **Flow Type** control to specify the type of power flow for which the sensitivities will be calculated. Finally, select the line/transformer, interface or bus.

Whenever you make a change to any of these settings, click **Calculate Sensitivities** to update the grid with the new sensitivities.

The bus columns (to add see [Display/Column Options](#)) of interest after this calculation include:

- *Sensitivity: Injection  $dValue/dP$*  gives the sensitivity of per unit voltage magnitude to an injection of real power at the bus chosen in the Select Device section.
- *Sensitivity: Injection  $dValue/dQ$*  gives the sensitivity of per unit voltage magnitude to an injection of reactive power at the bus chosen in the Select Device section.
- *Sensitivity: Injection  $dValue/dVsetpoint$  (for PV bus)* gives the sensitivity of per unit voltage magnitude to an injection of reactive power at the bus chosen in the Select Device section.
- *Sensitivity: Injection  $dAngle/dP$  (radians/MW)* gives the sensitivity of voltage angle magnitude to an injection of real power at the bus chosen in the Select Device section.
- *Sensitivity: Injection  $dAngle/dQ$  (radians/MVar)* gives the sensitivity of voltage angle to an injection of reactive power at the bus chosen in the Select Device section.

The option labeled **Set Out-Of-Service** allows you to approximate the sensitivities at out-of-service buses with the sensitivity of the closest bus. Otherwise the out-of-service buses will display sensitivities of 0 when the sensitivities are calculated.

### Single Transfer, Multiple Meters

The Single Meter, Multiple Transfers Tab of the Flows and Voltages Sensitivities Dialog show the effect of a single transfer of real or reactive power on all the voltages in the system. To specify the transfer choose the **Seller** and **Buyer** in the same manner done on the [Power Transfer Distribution Factors](#) dialog. The specify whether the calculation of Real Power (P), Reactive Power (Q), or both are of interest.

Click **Calculate Sensitivities** to perform the calculations.

After clicking **Calculate Sensitivities**, each row of the results gives the sensitivity of voltage at the respective bus to the power transfer defined by the seller and buyer. The bus columns (to add see [Display/Column Options](#)) of interest after this calculation include:

- *Sensitivity: Transfer  $dV/dP$*  gives the sensitivity of voltage to a real power transfer
- *Sensitivity: Transfer  $dV/dQ$*  gives the sensitivity of voltage to a reactive power transfer
- *Sensitivity: Transfer  $dAngle/dP$*  gives the sensitivity of voltage to a real power transfer
- *Sensitivity: Transfer  $dAngle/dQ$*  gives the sensitivity of voltage to a reactive power transfer

The check box option **Turn off generator AVR control for generators participating in transfer** is also available. Selecting this option will turn off AVR control for only those generators participating in the transfer with a non-zero participation factor, resolve the power flow, calculate the voltage sensitivities, turn the AVR control back on for those generators for which it was changed, and then resolve the power flow. Stated another way, this option assumes that all generators maintain a constant reactive power output and do not maintain their voltage setpoint.

### Self Sensitivity

The Self Sensitivity Tab of the Flows and Voltages Sensitivities Dialog shows the affect an of an injection of real or reactive power at a bus on its own voltage (assuming everything is absorbed at the island slack bus). It processes through each bus calculating its own self-sensitivity only.

Each row of the results gives the sensitivity of voltage at the respective bus to a power injection at the same bus.  $dV/dP$  gives the sensitivity of voltage to real power injection and  $dV/dQ$  gives the sensitivity of voltage to reactive power injection. To speed up the calculation, sensitivities are calculated only for the buses displayed in the Bus Sensitivities grid. The bus columns (to add see [Display/Column Options](#)) of interest after this calculation include:

- *Sensitivity: Self  $dV/dP$  (per unit/MW)* gives the sensitivity of voltage magnitude to an injection of real power.
- *Sensitivity: Self  $dV/dQ$  (per unit/MVar)* gives the sensitivity of voltage magnitude to an injection of reactive power
- *Sensitivity: Self  $dAngle/dP$  (radians/MW)* gives the sensitivity of voltage angle to an injection of real power.
- *Sensitivity: Self  $dAngle/dQ$  (radians/MVar)* gives the sensitivity of voltage angle to an injection of reactive power.

Click **Close** to close the Flows and Voltages Dialog.

---

## Loss Sensitivities

---

To access the Loss Sensitivities dialog go to the [Tools](#) ribbon tab, and choose **Sensitivities > Loss Sensitivities** from the [Run Mode](#) ribbon group. This is only available in [Run Mode](#).

The **Bus Marginal Loss Sensitivities Dialog** is used to calculate and display the sensitivity of a real power loss function,  $P_{\text{Losses}}$ , to bus real and reactive power injections. Stated mathematically, the display calculates  $d P_{\text{Losses}}/d P_i$  and  $d P_{\text{Losses}}/d Q_i$ , where  $P_i$  and  $Q_i$  are the real and reactive power injections at bus  $i$ , respectively.

Stated less formally, the display indicates how losses would change if one more MW or Mvar of power were injected at bus  $i$ . Simulator can calculate the losses for a bus relative to losses in the bus' island or area, to losses in a select group of areas, or, if the bus belongs to a super area, to losses in the bus' super area. How Simulator computes the losses is governed by the value of the **Loss Function Type** option.

The Loss Function Type may assume one of the following six values:

### Do Not Calculate Bus Loss Sensitivities

No Losses are calculated because a loss function is not specified.

### Each Electrical Island

Losses are calculated with respect to the losses in bus' island. If the power system consists of only one island, losses are computed with respect to the total system losses.

### Each Area

Losses are calculated with respect to the total losses for the area containing bus  $i$ . This is probably the most common loss function because usually one is concerned with minimizing losses for a particular area rather than for the entire case.

### Each Area or Super Area

Losses are calculated with respect to the total losses for the area containing bus  $i$  if bus  $i$  does not belong to a super area, and with respect to the total losses for the super area containing bus  $i$  if bus  $i$  does belong to a super area. When a case contains Super Areas this is more commonly done. When Simulator calculates loss sensitivities internally for use in the Economic Dispatch or the Optimal Power Flow calculation, this is the option which is used.

### Areas Selected on Loss Sensitivity Form

Losses are calculated with respect to the total losses for a group of areas, specified in the Selected Areas table.

### User-Specified

If you select User-Specified as the loss function type, the values last calculated using a different loss function type will become fixed. Thus you can force the loss sensitivities to remain constant when used in other features and tools of Simulator.

In steady-state power system operation, total generation must always equal total load plus losses. Therefore, the real power injection at a single bus cannot be changed arbitrarily; it must be met by a corresponding change somewhere else in the system so that the total power remains balanced. In other words, the change in power injection must somehow be absorbed. How the injection is absorbed depends on the Loss Function Type. If the Loss Function Type is **Each Island**, the injection is absorbed by the island slack. For the **Each Area** and **Selected Areas** loss functions, the injection is absorbed at the area tie-lines.

The loss sensitivities are calculated by modeling an injection of power at a bus and then assuming that this injection is absorbed by the island slack bus. The sensitivity then shows

how much the losses (for the region of interest) increase when you transfer 1 MW at the injection bus to the island slack. The "region of interest" is what was chosen as Island, Each Area, or Selected Areas.

Therefore, the "absolute numbers" given by the loss sensitivity dialogs are not directly meaningful because we are always assuming that the absorbing point is the island slack bus. What is meaningful is the "difference" between sensitivity numbers.

Example:

Assume the sensitivities are calculated to be

Bus A Loss MW Sensitivity = -0.04 = Asens

Bus B Loss MW Sensitivity = -0.02 = Bsens

Bus C Loss MW Sensitivity = +0.03 = Csens

Using these we can then look at the sensitivity of generic transfers between these buses by using "superposition".

Consider the following change in injections modeling a transfer of power from Bus A to Buses B and C.

Bus A injection = +10 MW = AMW

Bus B injection = - 6 MW = BMW

Bus C injection = - 4 MW = CMW

An estimate of the change in losses can then be calculated as

$$\begin{aligned}\text{Loss Change} &= (\text{AMW})(\text{Asens}) + (\text{BMW})(\text{Bsens}) + (\text{CMW})(\text{Csens}) \\ &= (+10)(-0.04) + (-6)(-0.02) + (-4)(+0.03) \\ &= -0.4 \text{ MW}\end{aligned}$$

The Bus Marginal Loss Sensitivities Dialog houses the following controls:

#### **Selected Areas Table**

This table is used only when the **Loss Function Type** is set to *Selected Areas*; otherwise, it is ignored. Left-click on the *Include* field to include or exclude areas from the loss function.

#### **Calculate Marginal Loss Sensitivities Button**

Once the loss function type has been specified, click this button to calculate the bus marginal loss sensitivities and update the Bus Marginal Loss Sensitivities table.

#### **Bus Marginal Loss Sensitivities Table**

This table shows the bus marginal loss sensitivities for all buses with valid [Area/Zone/Owner filters](#). The Bus Marginal Loss Sensitivities Table is a type of Case Information Display and thus exhibits features and behavior similar to all other case information displays. It has a local menu from which you can choose to find out more about a particular bus. You can sort records by any of the listed fields by clicking on the column headings. The table contains the following fields:

**Number, Name** Number and name of the bus.

**Area Number, Area Name** Number and name of the bus' area.

**Loss MW Sens.** Sensitivity of the loss function to an increase in the real power injection (generated power assumed positive) at the bus.

**Penalty Factors** Computed penalty factor of each bus.

**MVR Sens.** Sensitivity of the loss function to an increase in the reactive power injection (generated power assumed positive) at the bus.

**Just Generators Marginal Loss Sensitivities Table**

This table shows the bus marginal loss sensitivities for only the generator terminal buses with valid [Area/Zone/Owner filters](#). This table is otherwise identical to the displayed values and table operation as the Bus Marginal Loss Sensitivities Table described above.

---

## Transmission Loading Relief Sensitivities

---

Transmission Loading Relief (TLR) Sensitivities are similar to the [Power Transfer Distribution Factors](#). Both TLR Sensitivities and PTDFs measure the sensitivity of the flow on a device to a transaction. To calculate PTDFs, you specify a source group and a sink group, and Simulator determines the percentage of a single transfer between the source and sink that flows on each of several monitored elements. For TLR sensitivities, you specify a single device, such as a transmission line, to monitor, and a group that serves either as source or as sink. Simulator then determines the sensitivity of the flow on the single monitored element to many different transactions involving the group you specified as the source or sink. To summarize, PTDFs express the sensitivity of many monitored elements to a single transaction, whereas TLR sensitivities gauge the sensitivity of a single monitored element to many different power transfers.

"TLR" stands for "Transmission Loading Relief." TLR is an industry-wide tool for managing transmission utilization to prevent overload situations that put the system at risk. For example, suppose a particular line is loaded beyond its thermal limit. Its owner will request that a TLR program be initiated, which dictates that all transactions for which 5% or more of the exchanged power flows on the overloaded element be curtailed. The TLR sensitivity tool in Simulator is useful for pinpointing those transactions that would be curtailed. Suppose we use the TLR tool to determine where area A can purchase power from while the TLR for the overloaded element is in place. We specify area A as the buyer area, identify the overloaded line, and tell Simulator to perform the calculation. Simulator will then list the sensitivity of the flow on the overloaded line to power exchanges between all other generators, areas, and buses to area A. Any transaction for which the sensitivity exceeds 5% would be curtailed; anything below 5% would be allowed to continue.

To calculate TLR Sensitivities, select **Tools > Sensitivities > TLR Sensitivities** from the main menu. This will open the [TLR Sensitivities dialog](#), which will allow you to set TLR Sensitivities options and then calculate the sensitivities.



---

## TLR Sensitivities Dialog

---

The TLR Sensitivities dialog allows you to calculate [Transmission Loading Relief Sensitivities](#) for the load flow case at its solved load flow point. To access this dialog go to the [Tools](#) ribbon tab, and choose **Sensitivities > TLR Sensitivities / Generation Shift Factors** from the [Run Mode](#) ribbon group. This is only available in [Run Mode](#).

The following describes the sections of the dialog:

### Device Type

Select whether you want to calculate the sensitivities for a transmission line/transformer, for an interface, or for multiple elements. To calculate sensitivities for an interface, you must have the [interface defined](#) in the case.

When you choose an individual line/transformer or interface, you need to specify the device by selecting it from the list of devices. In this case you specify the From Bus, To Bus, and circuit identifier for a branch, or the interface name or number. When you select the Multiple Elements option, the device selection area of the dialog changes to allow you to select a [TLR Multiple Device Type](#). The TLR results display changes accordingly in order to accommodate TLR sensitivities for multiple elements.

### Line/XFMR or Interface Only: Choose device

Specify the From Bus, To Bus, and circuit identifier for a branch, or the interface name, number, and monitored flow direction.

For multiple elements, specify the multiple device type. Choices are: selected lines/transformers, selected interfaces, overloaded lines/transformers in the base case, overloaded interfaces in the base, overloaded lines/transformers during the set of contingencies, and overloaded interfaces during the set of contingencies.

### Multiple Elements Only: Select Lines/XFMRs

Opens the case information display for lines/transformers. To include a line/transformer in the TLR study, set the **Selected?** field to Yes.

### Multiple Elements Only: Select Interfaces

Opens the case information display for interfaces. To include an interface in the TLR study, set the **Selected?** field to Yes.

### Transactor Type

Specify if the sensitivities will be calculated for the transactor being the buyer or the seller.

### Transactor Object

Specify what the transactor will be. The choices are [Area](#), [Zone](#), [Super Area](#), [Slack](#), [Injection Group](#), and [Bus](#). When choosing Area, Zone, or Super Area, this really means all generators in the Area, Zone or Super Area weighted by the generator participation factor.

### TLR Sensitivities

Specify if the next set of calculated TLR sensitivities should replace the currently calculated values, or be appended to the current values. The normal option is to Clear before Calculate. When choosing to Append, then as new calculations are performed, the results will only replace the existing sensitivity value store if the new value is larger than the old value. (Note: in this situation, negative values are considered smaller than positive values).

### PTDF Calculation Method

Choose the solution method to use for calculating the sensitivities similar as done on the [PTDF Dialog](#).

Note: The Lossless DC methods will be affected by the **DC Power Flow Model** settings in [Power Flow Solution: DC Options](#). Click on the **DC Model Options** button to open the dialog which sets these options.

#### **Calculate TLR Sensitivities**

Click the button to calculate the sensitivities. The calculation processing through each bus and determines the sensitivity of the MW flow of selected device with respect to a change of real power at each bus. When the transactor type is buyer, then the assumption is that power is injected at each bus and absorbed at the transactor object. When the transactor type is seller, then the assumption is that power is injected at each transactor object and absorbed at each bus.

#### **DC Model Options**

Click this button to open the [Ref1351865574DC Options](#) page. Here you can adjust the DC options you want used if you are computing the PTDFs with a Lossless DC approximation.

#### **Include only AGCable Generators**

If checked, then only generators available for generation control will be included in the TLR sensitivity calculations.

#### **Set Sensitivities At Out-Of-Service Buses Equal to Closest**

Click this button to approximate the sensitivities at out-of-service buses with the sensitivity of the closest bus. Otherwise the out-of-service buses will display sensitivities of 0 when the sensitivities are calculated.

#### **Line/XFMR or Interface Only: Generator, Area, and Bus Sensitivities**

The Bus Sensitivities show the values calculated at each bus. Generator sensitivities show the same information but have only a list of generators with the P Sensitivity column showing the sensitivity calculated at the terminal bus. The Area Sensitivities calculated represent a weighted average of the generator buses in the area with the weighting done by generator participation factor.

#### **Multiple Elements Only: Mult. Bus Sensitivities and Gen Sensitivities**

The Bus Sensitivities show the values calculated at each bus. Generator sensitivities show the same information but have only a list of generators with the P Sensitivity column showing the sensitivity calculated at the terminal bus.

---

## TLR Multiple Device Type

---

The Multiple Device Type selector allows you to specify a group of elements for which Transmission Loading Relief (TLR) sensitivities will be calculated. The Multiple Device Type selection is accessible only when you chose the Multiple Elements option from the Device Type in the [TLR Sensitivity Dialog](#).

### Selected Devices

#### Select Lines/XFMRs

Select the Selected Devices option, and click on this button to open a dialog to choose the lines and transformers to be included in the calculation. Toggle the Selected field in the Lines and Transformer [Case Information Display](#) to Yes for the branches you wish to include. If the Selected field is not available in the information display, add that column to the display by right-clicking in the display and selecting [Display/Column Options](#). The ETLR of selected Lines/XFMRs is the algebraic sum of the TLRs of each individual element. The WTLR uses as weight the current MW flow in the element.

#### Select Interfaces

Select the Selected Devices option, and click on this button to open a dialog to choose the interfaces to be included in the calculation. Toggle the Selected field in the interfaces [Case Information Display](#) to Yes for the interfaces you wish to include. If the Selected field is not available in the information display, add that column to the display by right-clicking in the display and selecting [Display/Column Options](#). The ETLR of the selected interfaces is the sum of the TLRs of the interfaces that have a limit different from zero. The WTLR uses as weight the MW flow in the interface.

### Overloaded Devices

Select this option to include in the multiple element TLR calculation the branches and/or interfaces that are overloaded in the present case based on the [Limit Monitoring Settings and Limit Violations Dialog](#). The list of the overloaded branches and interfaces is available in the [Limit Monitoring Settings and Limit Violations](#) information display.

### CTG Overloaded Devices

Select this option to include in the multiple element TLR calculation the branches and/or interfaces that were identified as overloaded by the [Contingency Analysis](#) tool. A branch or interface is included in the calculation if it has been overloaded during at least one contingency. The list of branches and interfaces identified as overloaded can be accessed from the [Lines, Buses and Interfaces Tab](#) in the [Contingency Analysis Dialog](#).

The ETLR of the CTG overloaded lines and transformers is the algebraic sum of the individual TLRs. The WTLR uses as weight the Aggregate MVA Overload of each transmission line and transformer, which is defined as the sum of the MVA overload in the line or transformer across the contingencies that caused a violation in that particular line or transformer. The Aggregate MVA Overload and a related field, the Aggregate Percent Overload of a line or transformer, are measures of the weakness of that transmission line on the grid. The Aggregated MVA Overload and the Aggregate Percent Overload are line and transformer fields that can be displayed in the [Case Information Display](#), used in [Contouring](#), etc.

The ETLR of the CTG overloaded interfaces is the algebraic sum of the individual TLRs. The WTLR uses as weight the Aggregate MW Overload of each interface, which is defined as the sum of the MW overload of the interface across the contingencies that caused a violation in that interface. The Aggregate MW Overload and a related field, the Aggregate Percent

Overload of an interface are measures of the weakness of the interface. These two interface fields can be displayed in the [Case Information Display](#), used in [Contouring](#), etc.

#### **Effective Transmission Loading Relief (ETLR)**

In the same manner as the TLR represents the MW increase in an element per MW transfer, the ETLR column in the Mult. Bus Sensitivity table represents the total MW increase in all the elements in the set per MW increase of the transaction. Let us suppose that the set contains two transmission lines: A and B and assume that for a 1 MW transfer from bus i to the transactor, the flow in line A increases in 0.5 MW and the flow in line B decreases in -0.3. Then the ETLR of bus i is +0.2 since that is the total MW increase in the elements in the set. The ETLR provides a measure of the simultaneous MW change in multiple elements, and thus the overall effect in flows on the element of the set. The ETLR is a bus field that can be included in the bus [Case Information Display](#), used in [Contouring](#), etc.

#### **Weighted Transmission Loading Relief (WTLR)**

The WTLR column in the Mult. Bus Sensitivity table weights the sensitivities based on the flow or overload flow values of the element. It is a measure of the value of a certain bus to relief transmission loading. The buses with the highest WTLR (or lowest WTLR, depending on the transactor type) are identified as the most effective buses to mitigate transmission loading considering multiple elements. The WTLR is a bus field that can be included in the bus [Case Information Display](#), used in [Contouring](#), etc.

---

**Generation Shift Factor Sensitivities**

---

Generation Shift Factor (GSF) Sensitivities are a specific kind of TLR calculation. GSFs always involve a transfer with the slack bus being the Buyer. Other than this, GSF and TLR calculations are *identical*. See [TLR Sensitivities](#) for more information.

---

**Chapter 12: Time Step Simulation**

---

This chapter contains information on the new Time Step Simulation tool. This tool allows you to set up multiple solution runs with changing input data. This can be useful for running several OPF or SCOPF solutions in sequence, and storing information from each solution in tabular format.

This tool can also be handy as a training tool as well. As an instructor, you can set up a scenarios that the student can interact with, and have changes occur at prescribed intervals during their interaction.

- Time Step Simulation Overview
- Setup and Control
- Schedules
- Running the Simulation

---

## Time Step Simulation

---

The Time Step Simulation tool allows you to specify operating conditions and obtain power flow solutions for a set of points in time. It provides the tools needed to analyze the operation of a power system hour by hour.

Time Step Simulation is available in the base Simulator package. If you own Simulator, you can start taking advantage of this valuable tool right away. In addition, if you own Simulator OPF or SCOPF licenses, you can solve hourly OPF and SCOPF scenarios and use the tool to evaluate the behavior of prices and operating constraints. The tool will obtain the optimized generation dispatch for each hour of the analysis horizon.

In order to access the Time Step Simulation, go to the [Tools](#) ribbon tab, and choose **Time Step Simulation** from the [Run Mode](#) ribbon group. This is only available in [Run Mode](#). Please continue reading the following topics and take full advantage of this powerful tool.

[Time Step Simulation Quick Start](#)

[Time Step Simulation Dialog](#)

[Specifying and Maintaining a List of Time Points](#)

[Loading Hourly Input Data](#)

[Setting up Scheduled Input Data](#)

[Storing Input Data and Results](#)

[Running a Timed Simulation](#)

[Time Step Simulation Toolbar](#)

[Running OPF and SCOPF Time Step Simulations](#)

---

**Time Step Simulation: Quick Start**


---

The first time you access the Time Step Simulation by selecting **Time Step Simulation** from the [Run Mode](#) ribbon group on the [Tools](#) ribbon tab, you will see the [Time Step Simulation Dialog](#), which contains several pages. The **Hourly Summary** page is used to define and control the time points you want to analyze. As an example, we will assume that you have hourly load data for tomorrow and that you want to determine the system bus voltages for each hour. You would do the following:

**Step One: Set the List of Points**

In order to create a list of points, right click on the **Hourly Summary** page and select **Insert New Timepoint(s)**, which brings up the [New Timepoint Dialog](#). In this dialog, select tomorrow's date from the drop down calendar component. Set the field **Total Number of Timepoints to Enter** to 24. Assume the other default values and click OK. This will insert 24 timepoints one for each hour starting tomorrow at 1: 00 AM.

**Step Two: Specify Input Data**

Input data is specified in the **Input Page**. In this example your input data corresponds to hourly loads. Select the **Hourly MW Loads** sub-page. In order to specify hourly load values you have to insert a column for each load. Right Click on the grid and select **Scale/Insert Load Column(s)** to bring up the [Insert/Scale Column Dialog](#). In the selector component, select the load for which you want to specify hourly values. You can press the shift key to select multiple elements. Then press the blue **Arrow Button** to pass the selected loads to the right side of the selector. Select **OK** to insert the new column(s). Now you can specify the hourly MW values for those loads.

**Step Three: Specify the Custom Results**

Obtaining solutions for a large number of timepoints has the potential to create unnecessary burden in memory and storage due to the large amount of data that can be generated. For this reason, the Time Step Simulation Tool allows you to explicitly specify what quantities you want to display and store. This is done in the [Results Page](#). This page contains grids for many devices including Generators, Lines, etc. In this example we want to analyze the bus voltage magnitudes. We specify what quantities we want to store as results for each object by clicking the **View/Modify** button, which brings up the [Custom Results Selection Dialog](#). In this dialog select the **Buses** page and in the **Available Bus Fields** section check the **Per Unit Voltage** field. Results will be saved only for those buses that have the **Time Selected** field on the grid set to YES. Set this field to YES for the buses that you want to save. Click the **Save and Close** button to save the custom results settings, i.e., which objects, fields and records are to be kept during the solution; in this case bus voltages.

**Step Four: Run the Simulation**

The upper part of the Time Step Simulation form contains buttons used to control the simulation. To do a full run of the 24 hours click the **Do Run** button. The **Last Result** box shows the progress of the simulation as each time point is being solved. If you are in the **Results – Buses** page, you will see that a column was added for each bus set to YES in the **Custom Results Selection Dialog**. Each column shows the bus per unit voltage. Recall that you can **Right Click** on any Simulator grid and select **Plot Column** to obtain a plot of the column values. If you select cells spanning all the bus per unit voltage columns, you will obtain the voltage profiles for each bus, versus time.

If you are in the **Hourly Summary** page, the **Processed** column shows that each point was in fact processed and solved.

**Step Five: Save the Results**



Once you have completed the simulation, you can save the results in a **Time Series Binary** (tsb) file by pressing the **Save Data Binary** button. This file will contain all the input data, the simulation options, the custom results settings, and the results. You can reload this tsb file any time by pressing the **Read Data Binary** button.

---

**Time Step Simulation Dialog**


---

The Time Step Simulation Dialog is used to control and visualize the time simulation. The top section of the form contains buttons for data input/output and buttons to control the progress of the simulation. The main part of the form has a number of [Time Step Simulation Pages](#) that contain grids where input data can be specified and simulation results can be examined.

### Input/Output Buttons

#### Read TSB File

Press this button to read a Time Series Binary file (.tsb File). This file stores hourly and scheduled input data, the simulation options and the hourly results.

#### Read Load Format

Reads a .csv file containing hourly load MW and MVar values. The CSV format of this file is:

```
BUS_NUMBER,BUS_NAME,VST bus numbers,DPID,YrMoDay,Hr,KW,KV
446,ANAME,18803,506520,20041001,1,1970,769
446,ANAME,18803,506520,20041001,2,2005,821
```

#### Save TSB File

Press this button to save the hourly and scheduled input data, the simulation options, and the hourly results in a Time Series Binary file (tsb file).

#### Read Excel Data

Reads hourly load and generation data from Excel: The formats for generator and load data are:

##### Generators:

GEN DATA				
Date	Hour	BusNameGen1	BusNameGen2	...
Bus Numbers (s)		19306	100370	...
10/01/05	1	522	340	...
10/01/05	2	522	340	...
...				

##### Loads:

BUS_NUMBER	BUS_NAME	VST bus numbers	DPID	YrMoDay	Hr	KW	KV
2192	3BLUFF C	19306	100370	20050101	1	307	177
2192	3BLUFF C	19306	100370	20050101	2	311	-
							180
...							

### Simulation Control Buttons

During the solution, the Time Step Simulation solves each timepoint in a sequential manner. During this process, the Simulation can be in one of three states:

1. **Reset:** When the simulation has not started, when it has been completed, or when it has been paused and then reset.
2. **Running:** When the simulation is solving time points sequentially.
3. **Paused:** When the user has paused the simulation. The simulation actually waits until the present time point is solved in order to pause. If the simulation includes a SCOPF solution, then all the contingencies are processed and the system is optimized before the simulation is paused. Note that once a time point is solved its results are available on the various grids.

The user controls the Time Step Simulation by means of the following control buttons:

#### Starting Time

Use this selector to specify a start time point other than the first time point in the list. The simulation will disregard the time points before the selected start time point.

#### Ending Time

Use this selector to specify an end time point other than the end time point in the list. The simulation will stop right after the selected end time point.

#### Do Run [Pause Solution, Continue Solution]

Press this button to initialize the Time Step Simulation and go through all the timepoints until the last time point or a time point with Pause field set to YES is found. As the simulation takes place, the **Last Result Box** will be updated with messages. In addition, the [Hourly Summary Page](#) will change the Processed and Solved fields from No to Yes, reflecting the simulation progress. The result grids will be updated once the solution for a timepoint has been found.

When the Simulation starts, the simulation status changes to **Running**, and the **Do Run** button changes its caption to **Pause Solution**. Press this button to pause the Solution. The simulation will continue until the current time point is solved entirely. Once the Simulation status is set to **Paused**, the **Do Run** button caption changes to **Continue Solution**. Press this button to continue the solution.

#### Do Single Point

Use this button to solve the next time point. You can see the last processed point in the **Last Result** box. You can also select a specific starting point by using the **Start Time Point Selector**.

#### Reset Run

Use this button to go back to the first time point and initialize the simulation. This action does not delete the results of the time points processed so far, but it resets all the processed fields to No.

#### Last Result

The last result box in the lower left corner of the dialog is used to show solution progress messages. The messages indicate correct solutions or errors in the solution of the particular time point. It also shows the progress of the contingency analysis during a SCOPF solution.

#### Present Time

If you are solving the timepoints using a [scaled time delay](#), the Present Time of the process is displayed in this field.

---

**Time Step Simulation Toolbar**

---

The purpose of this toolbar (besides providing shortcuts) is to command a Time Step Simulation without having to keep the [Time Step Simulation Dialog](#) open. This is particularly important during a **Timed Simulation**, in which you want to see the changes on the online diagram as they occur in time.



The Figure shows the main functions of the toolbar. [Show/Hide the Time Simulation Form; Play/Pause/Reset/Next Time Control Buttons; Last Result Box; Present Time; Progress Bar, and Timed Simulation Options]. All the buttons, except the Show/Hide Time Simulation Form are disabled until a list of time points have been defined in the [Hourly Summary Page](#). The **Time Step Simulation Toolbar** is available in **Run Mode** and it is visible only when the [Time Step Simulation Dialog](#) is open.

---

**Time Step Simulation Pages**

---

The Time Step Simulation Dialog contains a number of pages used to specify hourly and scheduled input data, and to examine the results of the simulation. Please continue reading the following sections for a detailed explanation of each page:

[Hourly Summary Page](#)

[Input Page](#)

[Results Page](#)

[Results: Constraints Page](#)

[Options Page](#)

[TSB Description Page](#)

---

## Hourly Summary Page

---

The Hourly Summary Page of the Time Step Simulation Dialog is used to define the time points and to display a summary of the simulation. Most of the commands to manage time points can be accessed from the Hourly Summary Page Local Menu.

Once timepoints are defined, the Summary Page presents the following columns for each time point:

### Date

The date of the time point. To modify the date, Right Click and select **Change Timepoint Time** on the **Local Menu**.

### Hour

The time of the time point to the minute. To modify the time, Right Click and select **Change Timepoint Time**.

### Skip

This field is set to YES to include the time point in the Simulation. If set to NO, the hourly input data and any schedule action that occurs at this time point is not to the power system.

### Processed

This field is set to yes if the simulation has processed the time point. This includes applying the hourly input data, applying the scheduled actions and solving the power flow for that particular time point.

### Solution Type

The time point can be solved using one of the following **Solution Types**: Single Solution, Unconstrained OPF, optimal power flow (OPF), and security-constrained optimal power flow (SCOPF). The last three solution types require the Simulator OPF/SCOPF add-ons.

### Run Contingencies

If the case has a defined set of [contingencies](#), you can choose to run the contingency analysis for each time point by setting this option to YES.

### Solved

YES if a solution was obtained for the timepoint under the specified solution type.

### Num Loads

Total number of loads for which hourly input data has been specified.

### Total MW Load

Total MW load specified as hourly input data for the timepoint

### Total Mvar Load

Total reactive load specified as hourly input data for the timepoint

### Num Gens

Total number of generators in the system for which hourly input data has been specified

### Total MW Gen

Total MW of generation specified in the hourly input data.

### Pre Script Cmd

The Time Step Simulation has the capability of running a pre script command before each time point. The pre script command is run right before the time point is solved. Please check the [Script Command](#) section to learn the details about performing Simulator actions using script commands.

#### **Post Script Cmd**

The Time Step Simulation has the capability of running a post script command right after the time point is solved. Please check the [Script Command](#) section to learn the details about performing Simulator actions using the script language.

---

**Hourly Summary Page: Local Menu**

---

The Hourly Summary Page Local Menu is used to perform a number of logical actions on the timepoint grid and is, as every Simulator local menu, accessed by right clicking anywhere on the grid. When selected from the Hourly Summary Page, the local menu shows the following additional time step simulation specific options:

**Apply Time Point**

Simulator applies the hourly input data of that particular time point to the power system model, without solving the time point. Upon selection of this option, the timepoint information can be visualized in the [Case Information Displays](#).

**Solve Time Point**

This option applies the time point hourly and the scheduled input data to the power system, and solves the timepoint after the specified solution type.

**Change Time Point Time**

This option allows the user to modify the date time of the selected timepoint. If the new date time belongs to an existing point in the list, an error message is displayed. The timepoints can be specified with an accuracy of up to one minute.

**Display/Column Options**

Select this option to add or remove columns to the hourly summary grid.

**Set/Toggle/Columns > Select Column(s)**

This option is normally used to select the entire column. You can drag the mouse across several columns in a row, and use this option to select multiple columns. Then you can for instance copy the entire column data to the clipboard, or plot all the selected columns.

**Set/Toggle/Columns > Select Row(s)**

This options is normally used to select an entire row. You can drag the mouse across several rows within a column, and use this option to select multiple rows. Once the rows are selected, you can perform actions with the selected rows, such as copying data to the clipboard.

**Set/Toggle/Columns > Contour Column**

Use this option to contour a column in the data grid.

**Insert New Timepoint(s)**

Use this option to insert new timepoints into the timepoint list. This selection brings up the [New Timepoint Dialog](#).

**Set/Toggle/Columns > Plot Column(s)**

Use this option to automatically generate column plots. By default, in the Time Step Simulation grids, the column plots display graphs of column data versus the datetime column (the combination of the date and hour columns).

**Delete Entire Timepoint Record**

Use this option to delete the selected timepoint and all the hourly input data and custom results of that timepoint.



---

## Input Page

---

The Time Step Simulation Input Page is used to specify hourly input data and scheduled data. This requires that the list of timepoints have been created. Please read the [Specifying and Maintaining a List of Timepoints](#) for details on managing the list of time points.

The Input Page contains several pages that can be grouped in two types: Hourly Input Pages and Schedule Pages.

### Hourly Input Pages

The hourly input pages (Hourly MW Loads, Hourly Mvar Loads, Hourly Actual MW Generation, Hourly Maximum Generation, Hourly Line Status, Hourly Area Loads, and Hourly Zone Loads) are [Matrix Grids](#) that are used to specify data on an hour by hour manner. In order to tell Simulator that we want to specify hourly data for a load, generator, or line, we need to add that particular object to the corresponding grid. For instance, suppose that you want to specify hourly MW data for Load 1 at bus 1. Then go to the Hourly MW Loads page, right click and select **Insert/Scale Load Column(s)** to bring up the [Insert/Scale Column Dialog](#). In this dialog, you can select the load and add it as a column to the Load grid. The corresponding values for that particular load can then be entered.

A similar process is followed to add one or multiple hourly MW generation outputs or hourly line statuses.

Use the Hourly Area Loads and Hourly Zone Loads pages to specify the hourly MW load for an entire Area or Zone. When Simulator applies the input data to solve a timepoint, it will scale the load of the specified area or zone to match the value entered for that hour.

### Schedule Pages

The schedule pages ([Schedules Page](#) and [Sched Subscriptions Page](#)) are used to specify scheduled input data, i.e., data that does not follow an hour by hour format. Examples of data that can be scheduled are scheduled transactions, generator statuses, line statuses, etc. For a detailed explanation on how to set up scheduled data, please read the [Setting Up Scheduled Input Data](#) section.

---

## **Matrix Grids**

---

Matrix Grids are a special type of data grids used in the input and results pages of the Time Step Simulation tool. In these grids, the time dimension is assigned to the rows and the object fields (load MW, generator MW output, etc) are assigned to the columns. The column header corresponds to the ID of the object. The number of columns of the grid depends on the user options and simulation results:

### **Hourly Input Pages**

In the case of input pages, the user has to specify hourly data for each object, e.g., load, generator, area. Thus the user adds each column to the grid explicitly.

### **Custom Result Pages**

In the custom result pages, a column is created for each object whose results have been specified to be stored.

### **Constraint Pages**

Since the constraints are determined at solution time, the columns appear only when constraints have been detected during the solution.

---

**Results: Constraints Page**


---

This page is only available in OPF and SCOPF versions of Simulator. When the OPF solution type is used to solve a timepoint, the status of the system during that particular hour is optimized so that the total operating cost of the system is minimized and the normal operation constraints are enforced. In addition, the SCOPF solution type enforces contingency constraints. The OPF and SCOPF solutions contain information about the elements that determine the LMPs, the binding constraints, the violating contingencies, and changes in the control settings.

The information related to contingencies on these pages is available only in the SCOPF Simulator add-on.

The pages of the **Results: Constraints Page** are:

### Results Summary

This page is similar to the [Hourly Summary Page](#), except that it contains additional operating information such as:

**Initial Cost:** The cost of operating the system given the initial generator set points and a standard power flow solution. No controls are moved to minimize cost

**Unconstrained Cost:** Is the total operating cost after the controls are moved to minimize cost, without enforcing any normal operation or contingency constraint. The result is an operating cost that corresponds to an Economic Dispatch solution.

**Unconstrained LMP:** Is the average marginal price of the system under unconstrained optimization.

**Final Cost:** Is the total operating cost after a constrained (OPF or SCOPF) solution has been obtained.

**LMP (Average, Standard Deviation, Minimum and Maximum):** Metrics of the LMP values.

**Binding Lines:** Number of transmission lines and transformers that are binding after the OPF/SCOPF solution has been determined.

**# CTGs Unsolvable:** Number of unsolvable contingencies. These are severe contingencies that would cause the power flow solution to fail for that particular time point scenario. Unsolvability of the power flow case is related to maximum loadability conditions.

### Hourly Binding Lines

This is a [Matrix Grid](#) that shows information for transmission lines and transformers that become binding constraints during the OPF or SCOPF solutions. A similar page is available for binding interfaces.

When transmission lines or transformer thermal ratings become binding constraints for a timepoint, a column is automatically added forming in this manner the matrix grid. You can access the details of the binding line or transformer by right clicking and selecting [Show Binding Constraint Dialog](#) on the **Local Menu**. The grid shows also the following summary columns.

**Processed:** YES if the time point was correctly processed.

**CTGs with Viols:** Number of contingencies that presented one or more violations (violating contingencies)

**# CTGs Unsolvable:** Number of unsolvable contingencies for the timepoint

**BC Line Viols:** Number of transmission line and transformer violations that were identified in the base case.

**# Line Viol:** Number of transmission line and transformer thermal violations.

**Binding Lines:** Number of binding lines in the OPF/SCOPF solution.

**Line Unenforceable:** Number of lines with unenforceable limits.

### Hourly Binding Interfaces

This is a [Matrix Grid](#) that shows information for interfaces that become binding constraints during the OPF or SCOPF solutions. A similar page for lines is available.

When interfaces become binding constraints for a timepoint, a column is automatically added to the matrix grid. You can access the details of the binding interface by right clicking and selecting Show [Binding Constraint Dialog](#) on the **Local Menu**. The grid shows also the following summary columns.

**Processed:** YES if the time point was correctly processed.

**CTGs with Viols:** Number of contingencies that presented violations (violating contingencies)

**# CTGs Unsolveable:** Number of unsolvable contingencies for the timepoint

**BC Interface Viols:** Number of interface violations that were identified in the base case.

**# Interface Viol:** Number of interface violations.

**Binding Interface:** Number of binding interfaces in the OPF/SCOPF solution

**Interface Unenforceable:** Number of interfaces with unenforceable limits.

### Hourly Binding Contingencies

This [Matrix Grid](#) shows information similar to that found on the Hourly Binding Lines and Hourly Binding Interfaces pages. Here though the data is organized by contingencies, which allows easy identification of the most severe contingencies. Each column of the matrix grid corresponds to a contingency. The grid shows also the following summary columns.

**Processed:** YES if the time point was correctly processed.

**CTGs with Viols:** Number of contingencies that presented violations (violating contingencies)

**# Line Viol:** Number of transmission line and transformer thermal violations.

**Binding Lines:** Number of binding lines in the OPF/SCOPF solution

**Binding Interfaces:** Number of binding interfaces in the OPF/SCOPF solution

**Base Case:** Shows the number of violating contingencies in the base case.

### Binding Line Summary Matrix

This [Matrix Grid](#) shows the number of hours that a line has been binding due to each contingency. In this case, the rows correspond to binding lines, and the columns to violating contingencies. You can access the details of the binding element by right clicking and selecting Show [Binding Constraint Dialog](#) on the **Local Menu**. The grid shows also the following summary columns

**From Number:** Binding line from bus number

**To Number:** Binding line to bus number

**Circuit:** Binding line circuit ID

**Total Hrs:** Total number of hours the line was binding (in the overall simulation).

**Total Hrs Unenforceable:** Total number of hours the line constraint was unenforceable (in the overall simulation)

**Basecase:** Number of hours the line was binding in base case solutions.

### Binding Line Summary List

Shows information similar to the **Binding Line Summary Matrix** but in form of a List. The same line may appear several times under different contingencies. Besides the information on the **Binding Line Summary Matrix**, this grid shows the following fields.

**Contingency Name:** Contingency that causes the line constraint to be binding

**Avg MC:** Time Average MVA Marginal Cost

**Max MC:** Time Maximum MVA Marginal Cost

**Min MC:** Time Minimum MVA Marginal Cost

---

## Binding Elements Dialog

---

This dialog is called from the [Results: Constraints Page](#) of the [Time Step Simulation](#) dialog. Right-click on a time point record in the Results: Constraints table, and select **Show Dialog** from the popup menu. The dialog shows the binding line and interface constraints determined by the OPF/SCOPF solution. The **Time** selector allows easy navigation through the list of time points.

The grid section of the dialog shows the following:

**Type:** Either line or interface flow

**Constraint ID:** Line or Interface ID

**Contingency Name:** Name of the contingency under which the constraint becomes binding.

**MVA Marg. Cost.:** Marginal cost of enforcing the constraint. For lines, the limit corresponds to the thermal (MVA) limit. For interfaces, it is given by a MW limit.

---

## Results Page

---

The results of the Time Step Simulation are presented on the grids of the **Results Page**. The Time Step Simulation allows you to specify what objects (buses, lines, generators, loads, etc) and what object fields (bus voltage, bus LMP, gen MW, etc) should be displayed on the result grids. This gives the user the flexibility needed to explore the relevant results, avoiding at the same time the problem of storing a massive amount of results, most of which may not be relevant. Storage is a critical aspect of the Time Step Simulation, since a set of results comparable to full a PF/OPF/SCOPF solution is generated for each timepoint.

The **Results Page** has two sections: The top section is used to set up the options needed to customize the results display. The grid section is used to display the actual results.

### Results Page: Top Section

This section includes the following options:

#### View/Modify

Press this button to access the [Custom Results Selection Dialog](#). This dialog is used to specify the objects and object fields for which results will be stored.

#### Load

The custom result definitions set up in the [Custom Results Selection Dialog](#) can be saved in a **Results File** in order to use them with different **tsb files** or Simulator cases. Press this button to Load the Results File and apply the result definitions to the current Time Step Simulation.

#### Save

Press this button to save the result definitions in a **Results File**.

#### Group Results by

The Result Grids for Areas, Buses, etc are [Matrix Grids](#) that present the hourly results for each type of object. For instance, suppose that we want to store the bus voltage magnitude and angle. The Buses grid will show columns for the hourly values of voltage magnitude and voltage angle of each bus.

**Objects:** The columns will be grouped by objects, e.g., all the fields of bus 1, then all the fields of bus 2, etc.

**Fields:** The columns will be grouped by fields, e.g. all the bus p.u. voltages, then all the bus LMPs, etc.

#### Identify Results by

The Result Grids for Areas, Buses, etc. create one column for each object field specified in the [Custom Results Selection Dialog](#). The column header thus identifies the particular object for which hourly results are displayed. This identification can be made based on the object **Number**, **Name** or **Number + Name** combination.

### Results Page: Grid Section

This grid section is used to display the results of the [Custom Results Grid Pages](#).

---

## Custom Results Selection Dialog

---

This dialog is used to specify what objects and object fields will be stored during the Time Step Simulation and will be displayed on the [Results Grid Pages](#). This dialog is opened using the **View/Modify** button on the [Results](#) page of the Time Step Simulation dialog.

In order to tell the Time Step Simulation that you want to store a particular field for a certain object, you need to:

1. Click on the page of the object type you need to store (Area, Bus, etc)
2. Set to YES the **Time Selected** field of those objects for which you want to store information.
3. Check the boxes corresponding to those fields you want to store (bus LMP, bus p.u. voltage, etc)

Once you are done with the selections, press the **Save and Close** button to apply the customization. The [Custom Results Grid Pages](#) will be filled with the corresponding columns after the Time Step Simulation starts.

Note that the actual results and the result customization will be stored in the tsb file. For more information about saving the Time Step Simulation results, please read the [Storing Input Data and Results](#) section.

The following object's fields can be specified to be displayed on the grids and stored in the tsb file.

**Areas**

**Buses**

**Generators**

**Injection Groups**

**Interfaces**

**Lines**

**Owners**

**SuperAreas**

**Transformers**

**Zones**

To choose the fields for a specific type of object, click on the **Add/Remove Fields** button and select the fields from the list of available fields on the left side of the **Select Fields** dialog, and move them to the **Selected Fields** list on the right side of the dialog. Click OK to choose the selected fields.



---

## Results Grid Pages

---

These are the Pages of the [Results Page](#) that show the results of the Time Step Simulation after the results definitions specified in the [Custom Results Selection Dialog](#).

There is one page for each type of object: Areas, Buses, Generators, Injection Groups, Interfaces, Lines, Owner, Superareas, Transformers and Zones.

These pages are all [Matrix Grids](#). Each column of the grid corresponds to a field of a specific object of the power system, e.g., KV voltage of bus 1. Each row of the grid corresponds to a timepoint.

Note that you can plot the results of the grid versus the timepoint date time by selectiing **Plot Column(s)** on the **Local Menu**.

---

## Time Step Simulation Options

---

The Time Step Simulation is controlled through a set of options specified in the **Options Page**. The **Result Options** and the **Pricing Options** are available on this page only in the OPF/SCOPF Simulator add-ons.

### Area and Zone Load Scaling

In many practical studies, hourly data of individual MW load may not be available for all the loads in a control area. In order to simulate load variations, Simulator allows you to specify the hourly Area or Zone Total MW Load values in the **Hourly Area Loads** and **Hourly Zone Loads** page of the [Input Page](#). Thus the Time Step Simulation load data may be a combination of:

- Areas or zones where each individual load is specified
- Areas or zones where only the total hourly MW load is known.
- Areas or zones where some individual loads and the total area load are known.

If you have specified individual load values for a load within an area or zone that is also set to be scaled to a total MW value, Simulator will adjust the individual load to the individual value first, and then the load will be scaled with the rest of the loads in the area or zone to achieve the total MW value desired for the hour. The final load will be the total value specified for the area or zone.

When hourly values are specified for areas and zones within the same time step input, only one or the other can be used. This can be set in the **Use Hourly Values of** option.

You can also specify in the **Reactive Power Scaling** option how the reactive power is treated when the load is scaled. You can choose to have the reactive power scaled to keep the power factor constant, or for the reactive power to remain fixed at its original value.

### Solution Options

These options control the solution process.

**Pause if Power Flow Does Not Solve:** The Time Step Simulation is stopped at the timepoint where a power flow solution cannot be obtained. This is an indication of wrong data or a system brought to its loadability or transfer capability limit.

**Enable Power Flow Area Interchange:** This option ensures that the area interchange control, if possible, is enforced in the case.

**Use Parallel Contingency Analysis (on Dual CPU Machines):** Use this option to speed up the computation of the contingency analysis by distributing the processing time on several processors. This option is available for SCOPF.

### Pricing Options

These options are available only in the OPF/SCOPF add on.

**Solve Unconstrained Case:** Select this option when you want an unconstrained solution to be obtained before an OPF or SCOPF solution for each time point. When combined with the **Results Options** -> **Save Unconstrained Generator MW Outputs**, the MW Generator outputs obtained in the unconstrained solutions are saved.

**Price Hydro Generation at Marginal Cost:** During OPF and SCOPF simulation, hydro generation may experience large changes in output due to its low marginal cost. However, hydro generation is often not as cheap if limited water levels and dam restrictions are observed. These considerations are usually taken care of in the hydro-thermal coordination solution, outside of Simulator. In the OPF and SCOPF solutions it is important to assign a reasonable price to hydro generation to avoid large generation output deviations. A common

mechanism to do that is to first obtain the system marginal cost, and then assign this cost to the hydro units.

**Reset Hydro Generation Price at the End of Time Period:** Choose this option to make the hydro generation price be reset for the next time point solution.

**Save Binding Constraints:** The binding constraints determined in the OPF/SCOPF are stored.

### Time Step Simulation Options

The Time Step Simulation can be performed in two ways:

**Continuous:** In this case Simulator solves one time point after another immediately. The purpose of the simulation is to obtain the solutions for all time points as quickly as possible.

**Timed:** The purpose is to simulate the solutions as they would occur in actual time. In this case, the difference in date time between two time points in the list defines a delay to start the solution of the second point. A **Time Scale** is used to set the speed of the simulation with respect to actual time. Suppose that you have 3 time points defined at the following date times:

1/20/05 1:00 AM

1/20/05 2:00 AM

1/20/05 4:00 AM

Assume also that the **Time Scale** is 1 hour runs in 10 seconds. If you start the **Timed Simulation** you would see the conditions of the first time point applied to the power system immediately, the conditions of the second time point applied 10 seconds later, and those of the third point applied 20 seconds after the second point. The delays on the simulation allow you to see how the quantities evolve in actual time. In addition, you can animate the time simulation while each time point is being solved. The visualization of the **Timed Simulation** is enhanced when you use the [Time Step Simulation Toolbar](#).

**Step Type:** This section of options allows you to specify if the data for each time point should be applied only, or if the data should be applied and the load flow solved. You can also specify if the specified hourly data in the Input page should be applied, if schedules should be applied, and if the solution of the previous time step should be applied as the starting point for the next time step.

### Auto Contouring Options

The Time Step Simulation allows you to contour quantities on the online diagram using Simulator [Contouring](#) at each timepoint. Optionally, these contour diagrams can be saved in different formats.

**No Auto Contouring:** Contouring is not used during the Time Step Simulation. Although using the **Timed Simulation** you can see the quantities change and the animation take place on the diagrams, the contouring is not displayed.

**Contour but Do not Save:** Contouring takes place at each timepoint, but the diagrams are not saved.

**Save in File as Bitmap:** The contouring diagrams generated at each timepoint are saved in Bitmap format.

**Save in File as JPEG:** The contouring diagrams generated at each timepoint are saved in JPEG format.

**Contour File Name Format:** The Bitmap or JPEG contouring diagrams are saved using the specified format, which includes the timepoint date time.

### **Auto Load TSB File Options (saved in case pwb file)**

These are options that relate the power system case (.pwb File) to the time series binary file (.tsb file). These options are saved with the .pwb case.

**Automatically Load Default \*.tsb File:** The tsb file specified in the Default \*.tsb file is loaded automatically when opening the .pwb case. If the file cannot be found, a message will be issued.

**Automatically Run Simulation after Loading \*.tsb:** This option is only relevant if the tsb file was automatically loaded, and instructs the TSS form to solve the load flow immediately following loading the tsb file.

**Automatically Set Default \*.tsb File to Current \*.tsb File:** When leaving Simulator, the current time series information is saved in the Default \*.tsb file.

**Default \*.tsb File:** Default path and name of the \*.tsb file.

### **Run Pre-Script Command**

This option allows you to choose if script actions specified to be run before solving the load flow for certain hours should be implemented before or after specific input data designated for that hour is applied.

---

## Specifying and Maintaining a List of Timepoints

---

The list of timepoints is the basis for the Time Step Simulation. Simulator will go through the list of timepoints and solve each one of them in sequence. Results will be available only for those timepoints specified in the list.

Whether you start with an empty list or you already have time points in it, you can insert new time points by Right-Clicking in the **Hourly Summary** page and selecting **Insert New Timepoint(s)**. This will bring up the [New Time Point Dialog](#).

The list of timepoints will be always sorted based on the date time shown on the date and hour column in the [Hourly Summary Page](#). Thus, if you create a new time point with an intermediate date time, Simulator will insert it at the appropriate place in the list. If you need to change the date time of a timepoint you can **Right-Click** in the **Hourly Summary** grid and select **Change Timepoint Time** to bring up the [Change Timepoint Time Dialog](#). This dialog is similar to the [New Timepoint Dialog](#), with the exception that instead of specifying the date time for a new timepoint, you will be modifying the date time of an existing time point. If the date time matches the date time of an existing time point, a warning message is issued.

Each timepoint is linked to its hourly data in what is called a **Timepoint Record**. The **Timepoint Record** contains the date and time of the time point, all the input hourly data specified for that point, and if any, the results that have been obtained for that timepoint. If you delete a timepoint, the entire record is deleted with it. In order to delete the timepoint you can **Right Click** in the **Hourly Summary** grid and select **Delete Entire Timepoint Record**. Note that you can vertically select cells in this grid and delete several timepoints records at a time.

Suppose that you have a 1:00 AM and a 3:00 AM timepoints that have associated with them hourly MW load data. If you insert a 2:00 AM data, the 2:00 AM cell of the MW load will appear empty. If you run a study, no data will be applied to the power system at 2:00 AM, but you will get a result (identical to the one of 1:00 AM). You need to fill the 2:00 AM cells with data in order for the values to be applied at that timepoint and obtain the correct results for 2:00 AM.

---

## New Timepoint Dialog

---

The New Timepoint Dialog is used to define new timepoints. One or multiple timepoints can be defined at once. Use the date drop-down box to bring up a calendar for easy selection of the date. Note that the calendar has visual controls that allow you to navigate through months or years, which allows you to set the desired date easily.

If this is the first timepoint you will insert in the list, then by default the dialog is populated with today's date at 1:00 AM. If there are other points in the list, the default is one hour after the date time of the last time point.

The other options in this dialog are:

### Date

Use this control to select the date of the time point. You can use the visual controls or the up or down arrow keys to modify the date.

### Time

Use the control to enter the hour, minute and AM/PM description of the time. Once you are positioned on the hour, minute or AM/PM values, you can use the up/down arrow buttons for easy selection without having to type. Note that the timepoints are specified with an accuracy of up to minutes.

### Total Number of Timepoints to Enter

By default this field is set to one, meaning that you will enter only one timepoint with the specified date and time. If the value is more than one, several time points will be inserted in the list at the interval defined in the next two fields.

### New Timepoint Interval: Hours

If the **Total Number of Timepoints to Enter** is more than one, this field is used to specify the hour(s) of the interval between each time point. Timepoints will be Hours + Minutes + Seconds apart. Default is one hour.

### New Timepoint Interval: Minutes

If the **Total Number of Timepoints to Enter** is more than one, this field is used to specify the minute(s) of the interval between each time point. Timepoints will be Hours + Minutes + Seconds apart. Default is zero minutes.

### New Timepoint Interval: Seconds

If the **Total Number of Timepoints to Enter** is more than one, this field is used to specify the second(s) of the interval between each time point. Timepoints will be Hours + Minutes + Seconds apart. Default is zero minutes.

---

**Change Timepoint Time Dialog**

---

The Change Timepoint Time Dialog is used to modify the date time of a time point. Only one time point time can be changed at a time. If the new time assigned to the timepoint already exists, a warning message is generated asking to input a different date time. If the new date time does not exist, Simulator moves the time point to the correct position in the timepoint list.

**Date**

Use this control to select the date of the time point. Besides the visual controls, you can use the up or down arrow keys to modify the date.

**Time**

Use the control to enter the hour, minute, seconds and AM/PM description of the time. Once you are position on the hour, minute, seconds or AM/PM value, you can use the up/down buttons for easy selection without having to type. Note that the timepoints are specified with an accuracy of up to seconds.

---

**TSB Case Description Page**

---

This page is used to describe the time series binary file (tsb file) for information purposes. Simulator shows also the **Version Used to Store the TSB File** and the **Simulator Build Date**.



---

## Insert/Scale Column Dialog

---

The Insert/Scale Column Dialog is called from the [Input Page](#) grids. These [Matrix Grids](#) are used to specify hourly input data for Load MW, Load Mvar, Generator MW, Generator Maximum MW and Area Total MW Load. Hourly data of each particular object is specified in columns. When no column has been added to the grid, the Insert/Scale Column Dialog allows you only to select the new object. On the other hand, when there are existing columns on the grid, the Dialog allows you to insert a **New Column** either alone or based on the values of the existing columns. This option is available because values such as hourly load MW data tend to experience similar fluctuations in time.

**Current Column:** If there are exiting columns and the **Insert/Scale Column Dialog** is called from an hourly input data column, this field shows the position of that column. This field tells the user what column the **New Column** will be based on.

### Action:

These options tell how the new column will be inserted.

**Scale Entire Current Column:** This option is available only when the current column corresponds to an existing hourly data column. When applied, the new column takes the values of the current column scaled by the **Scaling Factor**.

**Scale Selected Rows of Current Column:** The Time Step Simulation allows you to select a group of contiguous rows of a column and scale only those rows of the **New Column**.

**Insert New Column Derived From Current Column:** This options copies the values of the **Current Column** to the **New Column**.

**Scaling Factor:** Factor used to scale the new column.

**Load Scaling:** These options are available only when inserting Load columns and are used to scale **Real and Reactive Load**, **Just Real Load**, or **Just Reactive Load**.

**New Columns:** Use this selector to specify the ID of the object that will be added as new column to the grid.

**Selector:** The bottom section of the dialog is a selector that allows you to specify one or multiple objects at a time and add columns for them by passing the objects from the left side to the right side. This selector has the following controls:

**Sort by Name:** Check this option to sort the list of available objects by name

**Sort by Number:** Check this option to sort the list of available objects by number

**Define Filter:** Press this button to filter the list of available objects using an [Advanced Filter](#).

**Use Area/Zone Filters:** Check this box to filter the list of available objects using the Area/Zone filters.

**Search Next:** By typing the start of the name of an object in the edit line, you can search the next object or all the objects available which match the search pattern. You can also use wildcards to search for objects.

**List of Objects:** The objects in the list are selected by clicking on them. Multiple objects that are together can be selected by clicking the mouse while holding the SHIFT key. Multiple objects that are not contiguous in the list can be selected by clicking the mouse while holding the CONTROL key.

**Select Button:** Press the blue arrow button to pass the selected objects on the left to the right.

**Remove Button:** Press the trash can button to remove the objects from the selected list on the right side.

For more information about methods to specify hourly input data, please read the [Loading Hourly Input Data](#) section.

---

## Loading Hourly Input Data

---

The Time Step Simulation allows you to specify the operating conditions of your power system through input data. The input data can be of two types:

**Scheduled Data**, which is specified for data that is not hourly by nature. To set up schedule data please read the [Setting up Schedule Input Data](#) section.

**Hourly Data**, or more precisely time point-based data, is specified for each time point in the [Input Pages](#). There are several ways to specify hourly input data:

- By entering data manually on the [Input Pages](#).
- By deriving or scaling values from another column using the [Insert/Scale Column Dialog](#).
- By loading previously formatted data from Excel or .csv files through the **Read Buttons** in the [Time Step Simulation Dialog](#).
- By pasting data from Excel directly to the grid. A common way to do that is to:
  - o Set up the desired timepoints and data columns on the grids of the [Input Pages](#)
  - o Copy the template to Excel by selecting **Copy** on the **Local Menu**
  - o Fill the Excel sheet with the appropriate data
  - o Paste the data back to the data grid by selecting **Paste** on the **Local Menu**

---

## Setting up Scheduled Input Data

---

There are two types of input data for the Time Step Simulation:

**Hourly Data**, which is described in the [Loading Hourly Input Data](#) section; and,

**Scheduled Data**, which is specified for data that is not hourly by nature. In this section we describe how to specify this type of input data.

Although it is possible to specify the operating conditions of a power system exclusively by hourly quantities, there are several quantities that are not hourly by nature, and whose hourly specification would be redundant and would require significant memory storage. Examples of such quantities are:

- The status of a transmission line that is taken out of service on a particular date and time for maintenance.
- The status of a generator, which follows a particular maintenance schedule.
- A generator's voltage set point that is different during the day or at night.
- A scheduled transaction between two areas that has different MW set points applied at 10 am, 4 pm and 10 pm.
- A capacitor connection status for a Mvar block that is used only during the day.
- An industrial load that operates at different levels for different shifts.
- A peaker generating unit that operates only during certain hours of the day

And many others. All these quantities can be specified by introducing the concept of **Schedule**. A schedule is a list of pairs (**Date Time, Value**), where the value can be numerical, conditional (Yes/No or Closed/Open) or text. The Schedule can have any number of time points and can be periodic. The schedule defines the "shape" of how a quantity varies in time. In order to define a schedule, go to the [Schedules Page](#) of the **Input Page**, and select **Insert New Schedule** to bring up the [Schedule Dialog](#). Please follow this link for a detailed explanation on how to define a schedule.

Once a schedule has been created, we can assign an object field, such as the status of a transmission line or the MW output of a generator to the schedule by means of a **Schedule Subscription**. The object field will follow the schedule "shape" in time. The use of Schedule and Schedule Subscription objects gives us great flexibility in specifying how quantities should vary. In particular, it is possible to assign many fields to the same schedule. In order to define a **Schedule Subscription**, go to the [Sched Subscriptions Page](#) of the **Input Page**, and select **Insert New Subscription** to bring up the [Sched Subscription Dialog](#). In this dialog you will encounter a detailed explanation of the Schedule Subscription use and capabilities.

An important feature of the Schedules is that their date times don't need to match the date times of the list of timepoints (time points listed in the Hourly Summary Page). Suppose that your list of time points are defined hourly for the next day: 1am, 2am, etc. up to 11pm. You can schedule a particular action to take place at 2:35 am and see the results of that action in the next time point, i.e., 3 am. A special logic takes care of applying scheduled action at appropriate time points asynchronously.

---

## Schedule Dialog

---

The Schedule Dialog is called from the [Schedules Page](#) and is used to define a schedule. A schedule is a list of time points together with a numeric, conditional (Yes/No or Closed/Open) or text value that defines the "shape" of how a quantity should vary in time. The timepoints are listed on the grid section of the dialog. The **Schedule Dialog** also contains a number of options that define the schedule.

When a schedule is created, the time step Simulation inserts by default four timepoints for the current date with a numeric value of zero in every point. You can then change the date and times of the schedule timepoints and specify different values.

### Schedule Dialog Local Menu Actions

The Schedule Dialog grid, as any other Simulator Grid has a Local Menu that is accessed by right-clicking on any cell. The most important options of this dialog are:

**Insert New Point(s):** Select this option to bring up the **New Schedule Point Dialog**. This is the same [New Time Point Dialog](#) but is used here to insert Schedule timepoints.

**Change Time(s):** Select this option if you want to change the date time of a schedule timepoint. The selection brings up the **Change Time Dialog**, which is the same [Change Timepoint Time Dialog](#) used here to change the date time of a schedule time point.

**Delete:** Use this option to delete the current schedule time point

**Select Column(s):** Use this option to select an entire column. Note that you can use this option and then delete all the timepoints.

**Copy:** Use this option to copy the values in the Schedule grid to the clipboard. If you are using Excel, you can modify the values and then Paste back the new schedule points into Simulator.

### Timepoint List Options

These options provide shortcuts to some of the **Local Menu** actions related to the maintenance of the list of Schedule time points:

**Add Point:** Adds a new timepoints at the end of the list. By default the date time of the new timepoint is one hour after the last time point. If there are no timepoints in the list, a timepoint is inserted with today's data at 1:00 AM.

**Delete Point:** Deletes the selected timepoint(s).

**Shift Date Time Buttons:** Use these buttons to move by one week, day, hour, minute or second the date time of the selected time point.

### Schedule Options

The upper part of the **Schedule Dialog** is used to define other Schedule Options:

**Schedule Name:** Enter here the name you want to give to the schedule. This is how you will identify the schedule when you set up schedule subscriptions. If the user does not specify otherwise, Simulator assigns names Sched1, Shced2, etc. as new schedules are created.

**Name Suffix:** This is a string automatically generated by Simulator, which is used to display the main characteristics of the Schedule together with the name. The Name Suffix is composed of three strings separated by points:

[**Num**, **Y/N** or **Txt**]: Indicates whether the schedule values are numeric, conditional or text.

**[NPER or PERxdyh]:** The string indicates whether the schedule is non-periodic (NPER) or periodic (PER). If it is periodic, it indicates the days (x) and hours (y) of the period.

**[Nn]:** Indicates the number of timepoints (n) of the Schedule.

By looking at the Schedule suffix, the user effectively avoids writing long names for the Schedule to characterize its properties.

**Value Type:** Numeric, conditional (Yes/No or Closed/Open) or Text. When you toggle this selection, the value column in the Schedule grid changes its heading from Numerical Value to Yes/No value to Text Value, and the numerical cells (blue) are changed to Yes/No cells (green), indicating that you can toggle the cell value, to blank for Text Value. Note that numeric fields, such as Load MW, Schedule Transactions MW etc, will subscribe to a Numeric Schedule, whereas fields such as Line Status will subscribe to a Yes/No Schedule. Text schedules can be subscribed to custom string and memo fields. For more details on Schedule Subscriptions, please read the [Sched Subscriptions Dialog](#).

### Periodic Options

A schedule becomes periodic when its values are repeated every certain period specified in days, hours, minutes and/or seconds. A logic condition for a schedule to be periodic is that its period be larger than the time span between the date time of the last time point and the first time point in the schedule. For instance, if the schedule has three time points at 1 am, 2 am and 4 am, the span is 3 hours. Thus the schedule period must be 4 hours or more. If this condition does not hold, Simulator issues a warning message.

**Repeat Every:** Check this box to make the schedule periodic. The schedule must have a total period greater than 0 to be considered periodic. The schedule period is the number of days plus the number of hours specified.

**Days:** A positive integer that specifies the number of days of the schedule period.

**Hours:** A positive integer from 0 to 23 that specifies the number of hours of the schedule period.

**Minutes:** A positive integer from 0 to 59 that specifies the number of minutes of the scheduled period.

**Seconds:** A positive integer from 0 to 59 that specifies the number of seconds of the scheduled period.

**Valid From:** When checked, this option sets a validity date for the schedule, whether it is periodic or not. No schedule actions will be applied to the power system before this date. By default, the Valid From date is set to Jan 01, 2000, at 12:00 am.

**Valid Until:** When checked, this option sets a validity date for the schedule, whether it is periodic or not. No schedule actions will be applied to the power system after this date. By default, the Valid Until date is set to Dec 31, 2030, at 12:00 am.

---

## Schedules Page

---

The **Schedules Page** is used to display all the defined schedules and their properties. The most relevant options of **Local Menu** of this page are:

### Schedules Page Local Menu Actions

**Insert New Schedule:** This option is used to define a new schedule through the [Schedule Dialog](#)

**Delete:** Use this option to delete the current Schedule.

**Show Dialog:** Select this option to bring up the Schedule Dialog with the information of the current Schedule.

### Schedule Page Fields

The Schedules Page shows the following information about schedules:

**Name:** Schedule Name

**Suffix:** Schedule Suffix. For a description on how the suffix is created, please see the [Schedule Dialog](#) section.

**Periodic:** Whether the schedule acts as periodic or not.

**Period Days:** Number of days in the schedule period.

**Period Hours:** Number of hours in the schedule period.

**Period Seconds:** Number of seconds in the schedule period.

**Valid From:** Whether a start validity date is used for the schedule.

**Period Start Date:** Start validity date.

**Period Start Hour:** Start validity time

**Valid Until:** Whether an end validity date is used for the schedule

**Period End Date:** End validity date

**Period End Hour:** End validity time

---

## Schedule Subscription Dialog

---

This dialog is used to define and display the options of a Schedule Subscription. Schedule Subscriptions tell the Time Step Simulation that a specific object field (Gen MW, Line Status, Scheduled Transaction MW, etc) should vary according to the "shape" specified in a Schedule.

Simulator objects have a large number of fields, which can be classified as numeric, Boolean, or text. Numeric fields, whether integer or real, can subscribe to Numeric Schedules, Boolean fields can subscribe to Yes/No Schedules, and text fields can subscribe to text schedules. Thus a condition for a field to subscribe to a schedule is that a schedule of its type be already defined.

The Dialog includes the following options:

**Active:** Indicates that the schedule subscription is active. If this box is not checked, the schedule subscription is defined, but no schedule action will be applied to the power system.

**Object Type:** Use this selector to specify the object type that you want to subscribe to a schedule. Currently, Simulator supports Generators, Loads, Line/Transformers, Shunts, Areas, Zones and Transactions. When you select an **Object Type**, three things happen:

- The **Object ID** list is populated with the elements of the object type present in the power flow case and the first element is displayed by default.
- The **Field** selector is populated with the enterable fields corresponding to that particular **Object Type** and the first field is displayed by default.
- The type of the first field (Numeric, Yes/No or text) is identified and the **Schedule** selector is populated with the existing schedules of that type.

**Object ID:** Use this selector to specify the particular object of a particular **Object Type** whose field you want to subscribe to a schedule. The **Object ID** has above it a string that indicates how the ID of the object is built depending on the **Object Type**. For instance, when the **Object Type** is Generator, the **Object ID** is specified as the Bus Number and then the Gen ID.

**Select/View Objects:** Press this button to bring up the Add Schedule Subscribers Dialog. This dialog is used to select multiple objects of the specified type, for which a field will subscribe to the schedule. When multiple objects are selected, the Object ID indicates "Multiple Objects" followed by the number of objects that will be selected. If there are objects whose fields are subscribed to a schedule, these objects will appear in the right side selection list, because they are currently subscribing to the schedule.

**Field:** Use this selector to indicate the particular **Field** that you want to subscribe to a schedule. Depending on the type, the Dialog indicates whether the **Field** should subscribe to a Numeric, Yes/No, or text schedule. Once you have selected the **Field**, the **Schedule** selector is populated only with the schedules of that particular type.

**Schedule:** Use this selector to specify the schedule to which the **Field** should be subscribed to.

**Time Shift:** The time shift options can be used to delay applying the scheduled actions by the specified **Delay Days**, **Delay Hours**, **Delay Minutes**, or **Delay Seconds**. For instance, suppose that you have setup a schedule so that a new 100MW generating unit enters online on March 10<sup>th</sup>. The entering schedule is complex and consists of increasing the output in steps of 20MW each hour starting at 10 am. Suppose that due to unexpected events, the connection of the unit must be put off by two days. Instead of modifying the entire schedule, you could set a 2 day (positive) delay so that the unit enters online on March 12<sup>th</sup> at 10 am.

**Subscription Type:** In addition to the time shift, the value "shape" defined by a numeric schedule can be altered by making the subscription **Relative**.



**Absolute:** Select this option so that the field takes the exact value specified in the schedule.

**Relative:** The numeric values of the schedule can be altered by a **Multiplier** and a **Value Shift**. In this linear modification, the value of the field that is applied to the power system will be equal to:

$$\text{Actual Field Value} = \text{Multiplier} * \text{Schedule Value} + \text{Value Shift}$$

---

## Schedule Subscriptions Page

---

The **Schedule Subscriptions Page** of the [Input Pages](#) is used to display all the schedule subscriptions and their properties.

### Schedule Subscriptions Page Local Menu Actions

**Insert New Subscription:** Use this option to define a new subscription through the [Schedule Subscription Dialog](#)

**Delete:** This option deletes the current Subscription.

**Show Dialog:** This option brings up the [Schedule Subscription Dialog](#) with the information of the current Schedule Subscription.

### Schedule Subscriptions Page Fields

The Schedule Subscriptions Page shows the following information:

**Object:** Object Type: Generator, Load, Line/Transformer, Shunts, Areas, Transactions

**Object IDs:** Are explained in the following Table:

Object Type	Object ID1 (Numeric)	Object ID2 Numeric	Object ID3 String[2]
Generator	Bus Number		Gen ID
Load	Bus Number		Load ID
Line/Transformer	From Bus Number	To Bus Number	Circuit ID
Shunt	Bus Number		Shunt ID
Area	Area Number		
Transaction	From Area Number	To Area Number	ID

**Object Field:** Field that subscribes to the schedule

**Schedule Name:** Name part of the schedule

**Suffix:** Suffix part of the Schedule Name

**Active:** If not active, then the schedule values are not applied to the power system

**Relative:** If not relative the field takes the exact schedule values. If relative the schedule takes the value of the schedule scaled by a multiplied and added a Value Shift.

**Multiplier:** Scaling factor used when the subscription is Relative.

**Value Shift:** Shift value used when the subscription is Relative.

**Delay/Advance?:** Indicates if the schedule should be delayed or advanced to a new point indicated by the shift values.

**Day Shift:** Number of days of the schedule time delay

**Hour Shift:** Number of hours of the schedule time delay

**Minute Shift:** Number of minutes of the schedule time delay

**Second Shift:** Number of seconds of the schedule time delay

---

## Running a Timed Simulation

---

The **Timed Simulation** is set up in the [Options Page](#) **Time Step Simulation Options** subsection.

By default, the **Do Run** button of the [Time Step Simulation Dialog](#) will run a **Continuous Simulation**, i.e., a simulation in which each time point is solved immediately after the previous time point. In this case the purpose of the Simulation is to obtain solution for the time points as fast as possible. As the solution progresses, the results for each hour in every hourly grid are refreshed, showing the user the evolution of the Simulation. Please read the [Time Step Simulation Quick Start](#) section for a quick introduction on how to run a **Continuous Simulation**.

On the other hand, the Time Step Simulation tool can also be used to run a **Timed Simulation**. In this case, the Simulation takes place according to a time scale proportional to the date times of the time points. When you click the **Do Run Button**, the Simulation progresses as if it was running in actual time. The simulation can also be paused and reset at any time by using the **Pause** and **Reset** Buttons of the [Time Step Simulation Dialog](#).

The following are some of the things you can do with the **Timed Simulation**:

- You can hide the [Time Step Simulation Dialog](#) and control the simulation by using the [Time Step Simulation Toolbar](#), which contains buttons such as **Do Run**, **Pause**, **Reset**, etc. which mimic the control buttons of the Dialog.
- You can visualize how the quantities vary proportionally to actual time after each time step is applied. Recall that the **Time Scale** defined in the [Options Page](#) indicates the relationship between the actual time and the time scale defined for the time points in Seconds per Hour. Thus a Time Scale value of 60 will indicate that one hour of time span between two time points will occur in 60 seconds of actual time in the **Timed Simulation**.
- You can also animate the flows of the solution of a time point while you wait for the solution of the next one.
- You can contour the online diagrams. And see how the visualization changes as the quantities vary in time, and in addition, you can save those contour diagrams as JPEG or bitmaps for each timepoint.
- Finally, you can act on the system (by closing capacitors, changing generator outputs, etc) before the next time point is applied simulate operating actions in response to system conditions.

All these actions can be combined in outstanding presentations to your colleagues or clients on how the system would evolve in time across different scenarios.

---

## Running OPF and SCOPF Time Step Simulations

---

Besides obtaining hourly power flow solutions for multiple time points, users that own the OPF and the SCOPF add-ons can obtain hourly optimal power flow and security-constrained optimal solutions. In the Hourly Summary Page, you can specify the following solution types:

- Power Flow
- Unconstrained Optimal Power Flow, which is equivalent to Economic Dispatch
- Optimal Power Flow (OPF)
- Security-Constrained Optimal Power Flow.

Note that different timepoints can be solved by any of the previously listed solution methods in the same **Time Step Simulation**. However, the solution settings of the previous timepoints are used as initial conditions for the solution of the next time point.

Simulator OPF and SCOPF tools are among the most advanced optimization packages for power systems. They have been extended in the latest versions of Simulator with many features, and have become complex analysis system. We recommend the user unfamiliar with OPF/SCOPF solutions to read the sections on [Optimal Power Flow](#) and [Security-Constrained Optimal Power Flow](#) before setting up OPF/SCOPF Time Step Simulations.

### Power Flow Time Step Simulation

The hourly power flow simulation allows the user to obtain AC or DC power flow solutions for a set of timepoints. During the Time Step Simulation all the power flow options defined in the [Simulator Options](#) page as well as in other dialogs are used for the solution. A key concept of the Time Step Simulation is that if you select **Solve Time Point** from the [Hourly Summary Page](#) or from the grids of the [Input Page](#), you would obtain the same solution that if you would hit the **Single Solution Button**. This is true, when the input data does not contain schedule data but only hourly input data. If your Simulation contains schedule data, there may have been scheduled actions that were applied in previous timepoints that are not being applied when you select the **Solve Time Point** option.

The power flow solution will observe all the power balance constraints, control limits and area interchange constrained defined in the power flow settings.

### Unconstrained OPF Time Step Simulation

In the unconstrained OPF solution, the Time Step Simulation removes all the constraints that would normally act in the OPF and optimizes the system to find the minimum operating cost settings. In doing so, Simulator will change the set points of the specified controls (generators and phase shifters) to minimize the cost of all Areas and Superareas set to OPF AGC control.

Besides the power flow solution options, the Unconstrained OPF simulation will take all the options that have been defined for a regular OPF solution. Most of these options are defined in the [OPF-Options Dialog](#), which is accessed using the **OPF Options and Results** button on the **Add Ons** Ribbon tab. All the settings such as objective function, cost of unenforceable constraints, control available, prices for controls, etc are defined in this dialog. Other options are defined for each particular object such as generators, loads, areas, lines, interfaces, etc.

### OPF Time Step Simulation

When using the OPF solution type, the Time Step Simulation applies the hourly and schedule input data and optimizes the control areas set to OPF to minimize cost while enforcing normal operation constraints: transmission line thermal limits, interface limits, generator control limits,

and load control limits. In doing so, the OPF algorithm detects the controls that need to be moved, the constraints that are binding at the solution point, and the unenforceable constraints, i.e., constraints that cannot be enforced with the available controls.

Some of the quantities that are of interest in the solution of the OPF algorithm are:

- **Unconstrained Generator MW Output.** Displayed in the Hourly Unconstraint Gen MW page of the [Input Page](#)
- **Final generator MW Output:** Optimal generator output at the solution points, displayed in the Hourly Final Generator MW Page
- **Change in Generator MW:** Difference between the unconstrained and constrained generator optimum settings, displayed in the Hourly Delta Gen MW Page.
- **Locational Marginal Prices:** These are displayed in the Hourly Final Bus LMP Page. Average LMP prices and other LMP metrics are also displayed in the [Results Page](#) for Areas, Injection Groups, Super Areas, and Zones.
- **Binding Constraints** as well as **Marginal Cost of Limit Enforcement** for lines and interfaces. These fields can be seen in the [Results: Constraints Page](#) and in the [Results Page](#) for Lines, Transformers, and Interfaces.
- **Unconstrained, Final and Congestion Cost,** displayed in the **Results Summary Page** for the entire system. These costs are also displayed for displayed for Areas, Owners, Superareas and Zones in the [Results Page](#).
- **LMP Profit** for Generators, Owners, and Zones are displayed in the corresponding grids of the [Results Page](#).

### SCOPF Time Step Simulation

The SCOPF combines the power of Simulator's OPF with the [Contingency Analysis Tool](#) to optimize a system for minimum cost while enforcing both normal operation and contingency constraints. At each time point, the SCOPF solution provides the optimal operation of the system so that if contingencies occur they would not create security violations. The locational marginal prices created in this manner are security-constrained signals to the market.

The solution of SCOPF Time Step Simulation depends on the options that have been set up for the following tools:

- Power Flow
- Optimal Power Flow
- Contingency Analysis
- Security Constrained Optimal Power Flow
- Time Domain OPF Options

The SCOPF Time Step Simulation does the following for each timepoint:

- Applies the hourly input data to the power system
- Applies scheduled actions determined by the schedule input data.
- Solves a power flow
- If specified, solves an unconstrained optimal power flow (economic dispatch)
- Initializes the base case of the security constrained OPF by solving a power flow or an OPF
- For the initialization system conditions, solves the list of contingencies
- Solves the SCOPF optimization problem: minimizes operating cost while enforcing normal and contingent constraints.
- Displays the results in all the result grids.

The SCOPF is on its own a complex computation that often requires significant computer resources. This is due mostly to the solution of the list of contingencies and the calculation of their sensitivities. The size of the problem can be dimensioned by:

- Size of the system, given by the number of buses and the areas to be optimized. This is difficult to reduce since the optimization problem is normally defined for a certain region.

- Number of contingencies, which can be reduced by developing a contingency screening using peak loading conditions.
- Number of constraints (monitored elements), which can be reduced by selecting critical element, e.g. interfaces and higher voltage transmission lines.
- Number of timepoints in the list.

Another mechanism to speed up the computation of the PF/OPF/SCOPF Time Step Simulation is to use DC solutions in some of the internal routines:

- AC or DC power flow
- AC or DC contingency analysis. This one will produce the larger time savings.
- AC or DC SCOPF

---

## Storing Input Data and Results

---

In large cases, the amount of data that can be potentially generated by a Time Step Simulation is significant since basically a full PF/OPF/SCOPF solution is available for each hour. A convenient method to store large amounts of data is to save the data in a binary file. The input data, both hourly and scheduled data, as well as the results can be stored in a **Time Series Binary File**, referred to in this help guide as the **.tsb file**.

Here is a summary of what will be stored in the **.tsb file** when you click the **Save Data Binary** button in the [Time Step Simulation Dialog](#):

- All hourly input data, defined in the [Input Pages](#) for hourly Load MW, Load Mvar, Generator MW, Generator Maximum MW, Area Total MW Load and Hourly Line Status.
- All scheduled input data, defined by the combination of Schedules and Schedule Subscriptions specified in the corresponding [Input Pages](#).
- The values displayed on the grids of the [Results: Constraints Page](#) for hourly Binding Lines, Binding Interfaces, Binding Contingencies, and the Binding Line Matrix and List.
- The customization settings defined in the pages of the [Custom Results Selection Dialog](#) for the hourly results of Areas, Buses, Generators, Injection Groups, Interfaces, Lines, Owner, Superareas, Transformers, and Zones.
- The customized results in the pages of the [Results Page](#) for hourly field values of Areas, Buses, Generators, Injection Groups, Interfaces, Lines, Owner, Superareas, Transformers, and Zones. Recall that many fields can be defined for each type of object.
- The options defined in the [Options Page](#), except the options set up in the **Auto Load TSB File Options**, which are saved with the **.pwb case**.
- The data of the last solution run contained in the [Hourly Summary Page](#): Skip, Processed and Solved fields.
- The **.tsb file** description

**Note:** Recall that the data of any Simulator grid can be copied to the clipboard and to Excel by selecting the options on the grid **Local Menu**.

---

## **Chapter 13: PowerWorld Simulator Add-On Tools**

---

This chapter contains information on the tools available for purchase for adding additional functionality to the Simulator base package.

- Voltage Adequacy and Stability Tool (PVQV)
- Optimal Power Flow (OPF)
- Security Constrained Optimal Power Flow (SCOPF)
- Available Transfer Capability Analysis (ATC)
- Simulator Automation Server (SIMAUTO)
- Simulator Automation Server (SIMAUTO) for Simulator version 9



---

## Introduction to Simulator Add-On Tools

---

In addition to the features of the base Simulator package, various add-on tools are available. A brief introduction to each follows:

### Voltage Adequacy and Stability Tool (PVQV)

The purpose of the PVQV add-on is to allow the user to analyze the voltage stability characteristics of a system. After the PVQV simulation is complete, the user can graph various system parameters. For more information, see the [PVQV Overview](#).

### Optimal Power Flow Tool (OPF)

The purpose of an OPF is to minimize an [objective \(or cost\) function](#). In Simulator OPF the Linear Programming OPF algorithm (LP OPF) determines the optimal solution by iterating between solving a standard power flow and solving a linear program to change the system controls thereby removing any limit violations. For more information see the [OPF Overview](#).

### Security Constrained Optimal Power Flow Tool (SCOPF)

The OPF tool minimizes an objective function (usually total operation cost) by changing different system controls while meeting power balance constraints and enforcing base case operating limits. The SCOPF tool takes it one step further by considering contingencies that may arise during system operation and ensuring that in addition to minimizing the objective function, no unmanageable contingency violations occur. For more information see the [SCOPF Overview](#).

### Available Transfer Capability Analysis Tool (ATC)

ATC analysis determines the maximum MW transfer possible between two parts of a power system without violating any limits. For more information see the [ATC Analysis Overview](#).

### Transient Stability (TS)

Transient Stability analysis allows the analysis of system dynamic response to a fault on the system. *The Transient Stability module has not yet been released with version 13.*

### PowerWorld Simulator Automation Server (SimAuto)

SimAuto provides PowerWorld customers the ability to access PowerWorld Simulator functionality within a program written externally by the user. The Simulator Automation Server acts as a COM object, which can be accessed from various programming languages that have COM compatibility. Examples of programming tools with COM compatibility are Borland® Delphi, Microsoft® Visual C++, Microsoft® Visual Basic, and Matlab® (among others). For more information on SimAuto, see the [SimAuto Overview](#).

---

## PowerWorld Simulator PV/QV Overview

---

The PVQV tool is only available if you have purchased the PVQV add-on to the base Simulator package. [Contact PowerWorld Corporation](#) for details about ordering the PVQV version of Simulator.

PVQV, PowerWorld's voltage adequacy and stability assessment tool, is used to analyze the voltage characteristics of a power system.

PowerWorld Corporation also offers Optimal Power Flow ([OPF](#)), Available Transfer Capability [ATC](#), Simulation Automation Server ([SimAuto](#)), and Security Constrained Optimal Power Flow ([SCOPE](#)) add-ons. For more information see .

The PowerWorld Simulator (Simulator) is an interactive power system simulation package designed to simulate high voltage power system operation. In the base package, Simulator solves the power flow equations using a Newton-Raphson power flow algorithm. However, with the voltage adequacy and stability tool (**PVQV**) add-on the user can solve multiple power flow solutions in order to generate a [PV curve](#) for a particular transfer or a [QV curve](#) at a given bus.

The PVQV functionality is accessed using the PVQV main menu item. The commands available from this menu are [Refine Model](#), [PV Curves](#), and [QV Curves](#).

The purpose of the PVQV add-on is to allow the user to monitor any system parameter while automatically increasing a user-defined transfer. The PVQV module uses the Simulator built-in Newton Raphson power flow algorithm to accomplish this task. After the PVQV simulation is completed, the user can choose to graph any of the [Monitored](#) system parameters, designated in [Quantities to Track](#).

---

**PV/QV PV Curves**

---

The voltage stability function of Simulator provides the ability to compute PV curves for any bus in the system. Select **PV Curves** on the **Add Ons** ribbon group to open the [PV Curve dialog](#). The PV Curve dialog allows you to specify the elements to be tracked, set defaults for the PV curves, and run the PV analysis.

---

## **PV/QV PV Curve Dialog**

---

To display this dialog, go to the [Add Ons](#) ribbon tab and select **PV Curves** from the **PVQV** ribbon group.

The integrated PV / QV PV dialog contains all of the setup and controls for processing and analyzing the PV and QV curve analysis. The dialog is broken down into several pages:

[Setup](#)

[Quantities to track](#)

[Limit violations](#)

[PV output](#)

[QV setup](#)

[PV results](#)

---

## PV/QV Setup

---

The first step in the setup process is to define the source and the sink for the study transaction. The PV/QV tool expects the source and the sink to be [injection groups](#) defined by the user. If injection groups have been previously created, they can be selected by clicking on the drop-down list arrow of the Source and Sink boxes. If the injection groups have not been previously defined in the current case, they can be created by clicking the **View / Define Groups** button, right-clicking on the resulting list display, and choosing **Insert** from the popup menu. Alternatively, if a list of injection groups has been previously saved in a Simulator [Auxiliary](#) file, they can be loaded into the current case by again clicking the **View / Define Groups** button, right-clicking on the injection group list display, and choosing **Load > Auxiliary File** from the popup menu.

Once the source and sink points have been defined, the PV study will model an increasing transfer of power from these source points to the sink points. The transfer process is performed incrementally, based upon user specified [options](#) on how the transfer should vary during the solution process.

Note that options can be saved to or loaded from an auxiliary file using the **Save options** and **Load options** buttons, located at the bottom of the display.

### Manage contingency list

Clicking this button will open the [contingency analysis dialog](#) for managing or inserting contingencies to be processed during the PV/QV analysis. Contingencies defined and marked for processing in the contingency analysis dialog will be included.

To globally omit the inclusion of contingencies during the PV/QV analysis, check the box labeled **Skip Contingencies**.

### Base Case Solution Options

Clicking this button will open the [Simulator Options Dialog](#), allowing the customization of the solution options prior to processing the PV/QV analysis.

### Run base case to completion

Check this box if you wish to find the critical transfer point of the base case condition, in addition to the specified number of critical cases for the defined contingency scenarios. By default, the PV analysis will process until the number of critical cases (or scenarios) as specified on the [PV Results](#) page have been found, and will halt the process at that point. If the base case transfer scenario is not one of the critical cases, then checking this option indicates that you want the PV analysis to continue incrementing transfers for the base case condition to find the critical point of the base case, IN ADDITION to the number of critical cases specified.

The remainder of the setup options are found under the [PV/QV Options to Control the Transfer Increase](#).

---

## PV/QV Options to Control the Transfer Increase

---

These options are located on the [Setup](#) tab of the PV Study Form.

### Source

Use this dropdown box to identify the source [injection group](#). To model an increase in transfer, generator points in the source injection group will increase their output, and load points will decrease their magnitude in amounts proportional to their participation factors.

### Sink

Use this dropdown box to identify the sink [injection group](#). To model an increase in transfer, generator points in the sink injection group will decrease their output, and load points will increase their magnitude in amounts proportional to their participation factors.

### Initial Step Size (MW)

This option indicates the initial rate at which the transfer will be increased following each successful iteration. The default value is 100 MW. This tells Simulator to begin studying the transaction in 100 MW increments.

### Minimum Step Size (MW)

Whenever Simulator fails to solve the system at a given transfer level, it will return to the previously solved transfer level, reduce the step size by specified factor, and then try to solve the system with the transfer incremented by the newly reduced step size. The **Minimum Step Size** option specifies the minimum size this increment can be. Once the system fails to solve when the step size is at this value, Simulator will conclude that we have come very close to the voltage collapse point and terminate the analysis. So, the minimum step size essentially functions as a tolerance for computing the voltage collapse point. The default value is 10 MW.

### When convergence fails, reduce step by a factor of...

Whenever Simulator fails to solve the system at a given transfer level, it will reduce the transfer step size by the value specified for this option. The default value is 2. Therefore, Simulator will start incrementing the transfer in 100 MW steps. When it reaches a transfer level that it cannot solve, it will return to the last solved transfer level, reduce the step size to 50 MW, increment the transfer by 50 MW, and attempt to solve the case again. The next time it fails to solve, it will reduce the step size to 25 MW, and then to 12.5 MW, and finally to 6.25 MW. Since 6.25 MW is less than the **Minimum Step Size** value of 10 MW, it will instead use a final step size of 10 MW. Once the system fails to converge with this step size, the analysis will terminate, since it will conclude that it has arrived at the voltage collapse point, within the specified tolerance.

### Stop when transfer exceeds

Provide a MW transfer limit between the source and the sink. When the PV analysis reaches a transfer amount equal to this value, the PV analysis will terminate.

### How should reactive power load change as real power load is ramped?

This option controls how reactive load should vary as real load is changed during the analysis. Selecting **Maintain the MW/MVAR ratio at each load** will maintain the same ratio of real to reactive power load prior to starting the analysis. Selecting **Maintain the MW/MVAR ratio at each load, but then scale MVAR by a factor of** will maintain the same ratio of real to reactive power load prior to starting the analysis but will then scale the Mvar load by the specified factor. To change the reactive load by a specified power factor,

select **As MW changes, change the MVAR at a power factor of** and enter the power factor.

### Proportions for load components

When the option to **Vary Active Load Components by Proportion** is checked, the proportions for constant power, constant current, and constant impedance loads can be specified for both the source and the sink. The total load change is then distributed among the components based on their specified proportion.

### Allow only AGC units to vary

Control areas in Simulator may practice some type of [automatic generation control](#). When an area is on AGC control, all generators in the area that are, in fact, AGC-able will participate in the area's automatic generation control program. Thus, Simulator distinguishes individual generating units according to whether they do or do not participate in their area's AGC effort. By checking this option, Simulator will allow only those generators that are eligible to participate in AGC to contribute to the power transfer being studied. Otherwise, all units in the injection groups will be allowed to participate regardless of their AGC status.

### Enforce unit MW limits

If this option is checked, the output of any participating generating unit will be kept within its designed operating range of  $\text{MinMW} < \text{Output} < \text{MaxMW}$ . When a unit is pegged at one of its limits, participation factors of the other points in the limited generator's injection group will be adjusted to pick up the difference.

### Do not allow negative loads

This is the analog of the previous option for loads. If a load is used as a **source** point, it will be decreased to make power available for the transfer. Checking this option will instruct Simulator to keep loads from falling below 0 MW. If a particular load is capped at 0 MW, participation factors for the remaining points in its injection group will be recalculated to make up the difference. This option is irrelevant if loads are not being used as source points.

### Dispatch generators in merit order

When choosing to dispatch in merit order, injection groups with generators will be dispatched by moving individual generators to their maximum/minimum outputs in succession based on their relative participation factors.

### Skip contingencies

The PV Curve tool computes PV curves for both the base topology and for any contingencies that have been defined, unless the **Skip contingencies** checkbox is checked. If the **Skip contingencies** checkbox is checked, a PV curve will be computed only for the model in its present topology.

### Manage contingency list...

Click the **Manage contingency list...** button to open the [contingency analysis dialog](#). This will enable the creation, modification, and removal of contingencies from the list of configurations the PV curve tool will process.

### Run base case to completion

The PV curve tool is designed to ramp a transfer until the prescribed number of unsolvable cases, including both unsolvable contingencies and an unsolvable base topology, have been found. If the requested number of unsolvable cases have all been identified as being associated with contingencies, the tool will not reveal how much a transfer can be ramped for the base topology, unless the **Run base case to completion** checkbox is checked. Checking this checkbox forces the tool to continue to ramp the transfer until the base case

can no longer be solved, regardless of whether the requested number of unsolvable transfer level / topology combinations have been found.

**Base Case Solution Options ...**

Click this button to bring up the [Simulator Options Dialog](#). This allows the specification of the solution options to use for solving pre-contingency cases.



---

## PV/QV Quantities to Track

---

This section of the PV/QV dialog allows the definition of quantities to monitor (store) as the transfer is increased. Any unselected system parameters will not be saved. The **Quantities to Track** page contains several sub-tabs that allow the monitoring of different types of objects including: buses, generators, injection groups, lines, transformers, shunts and interfaces. There is also a sub-tab for monitoring **Devices at Limits**.

### Buses

For buses, the following can be monitored during the PV analysis: voltages, angles, MW load, Mvar load, shunts, the sensitivity of the voltage to changes in reactive power ( $dV/dQ$ ), and the sensitivity of voltage changes to the transfer (VP Sensitivity). In addition, whether or not a QV curve should be generated for the bus can be specified. The default value of all bus quantities is *NO*, indicating that the analysis will not monitor any bus-related quantities. To monitor a particular quantity, double-click the corresponding value to toggle it to *YES*.

### Generators

For generators, the following can be monitored: MW output, Mvar output, and Mvar reserve. The display operates exactly like the bus display.

### Groups

The **Groups** sub-tab allows the monitoring of the total generator MW and Mvar for the group, the total load MW and MVAR, and the Mvar reserve for the group. This display operates like the bus display.

### Lines

Note: all branches (whether transmission lines or transformers) appear on the **Lines** sub-tab. The **Lines** sub-tab allows the monitoring of real, reactive, and MVA flow (in either the FROM-TO or the TO-FROM direction), the MW and Mvar losses, and the [PTDF](#) value on any branch. For the flow fields, double-clicking on a particular entry will toggle its value from *NO* to *FROM-TO*, and double-clicking again will toggle its value to *TO-FROM*. Double-clicking the losses or PTDF value field will toggle the value between *YES* and *NO*. If the branch contains a transformer, the XFMR Tap can also be monitored.

### Transformers

Note: all branches (whether transmission lines or transformers) appear on the **Lines** sub-tab. See **Lines** above for setting up common branch quantities. For branches containing transformers, the **Transformers** sub-tab is used to set the transformer type (Fixed, LTC, Mvar or Phase) and set monitoring for Regulated Value, Tap Position and Regulation Error. If no transformers exist in the case, this option cannot be used.

### Shunts

The **Shunts** sub-tab allows the monitoring of Actual and Nominal Real(P) and Reactive(Q) Power, as well as the Regulation Error and Regulated Value. Double-click the value fields to toggle between *YES* and *NO*. If no shunts exist in the case, this option cannot be used.

### Interfaces

Simulator allows the definition of groups of branches that together comprise an [Interface](#). The **Interfaces** sub-tab allows the monitoring of real, reactive, and MVA flow, the MW and Mvar losses, and the [PTDF](#) value on any interface that has been defined. This option cannot be used if no interfaces have been defined.

### Devices at Limits

The **Devices at Limits** sub-tab allows the selection of various options to track devices that hit or back off limits during the PV analysis. Generators and switched shunts at var limits, LTC transformers at tap limits, and lines and interfaces at thermal limits can all be tracked. To track the limits of any of these devices, check the appropriate checkbox. Next to each checkbox there is a dropdown box for selecting an [advanced filter](#) to apply to the tracking. A new filter can be defined for a particular device by clicking the **Define** button next to that device. The filter will limit the number of devices that get tracked. It is a good idea to define a filter for tracking devices so that all elements in the case will not be tracked. Keep in mind that tracking any quantities will take up space in the computer memory. If too many devices are tracked, it is possible to run out of memory.

---

## PV/QV Limit Violations

---

The Limit Violations tab allows the definition of what should be considered a violation of a monitored voltage.

### Identify bus voltages

The **Low Voltage** and **High Voltage** options should be checked in order to see which of the buses are violating their voltage limits at the end of the PV/QV analysis. The limits used are those defined using the [limit monitoring settings](#). To examine the limit monitoring settings, press the **Limit Group Definitions** button to open the [Limit Monitoring Dialog](#).

When choosing the identify limit violations, the Overview table on the [Results](#) tab will contain information about the number of violations (#Viol), worst voltage violation (Worst V Viol), and bus at which the worst voltage violation occurs (Worst V Bus). Also, for any scenario in which a violation occurs, the **Show Violations** option is enabled on the local right-click menu of the Overview table. Choosing this option will open a table detailing all of the violations.

### Inadequate voltage level

This option allows the specification of the value at which a monitored voltage is determined to be "inadequate". The results of the PV/QV analysis will report the transfer level at which the first instance of a voltage below the inadequate level was detected. There are three options for specifying the inadequate voltage. A voltage can be specified by selecting **Specify voltage for all buses in pu** and then entering a voltage. Low voltage limits can be specified with each bus. To use the values specified with each bus, select **Use Low Voltage Violation Limits for each bus**. Different sets of limits can also be specified with each bus. To use a particular limit set, select **Use a specified Low Voltage Limit Set** and select the limit set from the dropdown box.

The PV/QV analysis can be halted when it first detects the inadequate voltage. Checking **Stop when voltage becomes inadequate** will force the PV/QV analysis to do so.

To ignore the inadequate voltage feature of the PV/QV analysis, uncheck the **Identify inadequate voltage** checkbox.

---

**PV/QV PV Output**


---

Simulator records the value of each monitored quantity at each transfer level. However, unless you tell Simulator where to write the data, it will be present only in memory. The **Output Tab** allows you to designate where the data should be logged.

Click the **Save Results to File** option to indicate that you want to write results to a file. Then, in the adjacent text box, supply the complete path for the output file. You may use the **Browse** button to locate the place where you want the file to be written. The results can either be saved as a text file with .TXT extension or a comma-separated variable file with .CSV extension. The file type should be set in the save dialog under the Save as type option. If saving results to file, a log file will also be created that indicates how long the entire PV curve process took and the different base case and contingency scenarios examined at each nominal shift level. If running multiple instances of Simulator for PV analysis, a unique file name must be given to each set of results to avoid overwriting existing results.

Two file formats are available when saving results to file. If choosing to use a **Single Header File**, only a single field header will be shown at the top of the file. The first line of the file indicates that these are the results for a PV study. Following the header are lines indicating scenarios that have become critical during the analysis. Following this information is the single header line that indicates what fields are provided with the results. Following the header are the results for each scenario. An example is given below:

**Example Single Header File Output (CSV):**

```
RESULTS FOR PV STUDY "PVSTUDY"
"Newly Critical","L_00002Two-00006SixC1 (Reached Nose)"
"Scenario","Export","Import","Bus 3 (Three_138.0) PU Volt","Bus 5 (Five_138.0) PU Volt"
base case, 0.0000, 0.0000, 0.9927, 1.0066
base case, 100.0000, -98.7366, 0.9928, 1.0062
base case, 200.0000,-193.4267, 0.9928, 1.0054
base case, 300.0000,-278.6272, 0.9928, 1.0043
base case, 400.0000,-365.9344, 0.9927, 1.0031
base case, 500.0000,-449.7524, 0.9926, 1.0015
base case, 599.9999,-525.2494, 0.9924, 0.9996
base case, 650.0000,-563.8935, 0.9923, 0.9986
base case, 674.9999,-582.8201, 0.9923, 0.9980
L_00001One-00002TwoC1, 0.0000, 0.0000, 0.9906, 1.0067
L_00001One-00002TwoC1, 100.0000, -98.7366, 0.9882, 1.0062
L_00001One-00002TwoC1, 200.0000,-193.4267, 0.9849, 1.0053
```

If using the default output format and not using the single header file, each scenario will be written as a section with a header line at the beginning of each section indicating what fields are included. Scenarios that become critical during the analysis will only be listed with the base case scenario and will be listed at the shift level at which they became critical. An example of this file format is given below:

**Example for Default PV Output (CSV):**

RESULTS FOR PV STUDY "PVSTUDY" \*

\*Scenario\*\* "base case"

"Export", "Import", "Bus 3 (Three\_138.0) PU Volt", "Bus 5 (Five\_138.0) PU Volt", "Newly Critical"  
 0.0000, 0.0000, 0.9927, 1.0066,  
 100.0000, -98.7365, 0.9928, 1.0062,  
 200.0000, -193.4265, 0.9928, 1.0054,  
 300.0000, -278.6270, 0.9928, 1.0043,  
 400.0000, -365.9341, 0.9927, 1.0031,  
 500.0000, -449.7520, 0.9926, 1.0015,  
 599.9999, -525.2490, 0.9924, 0.9996,  
 650.0000, -563.8931, 0.9923, 0.9986,  
 674.9999, -582.8196, 0.9923, 0.9980, L\_00002Two-00006SixC1 (Reached Nose)

In addition to recording how the monitored quantities vary with the transfer, you can instruct Simulator to save the entire system state at regular intervals during the analysis. This can be helpful if you want to analyze particular transfer levels more closely after the analysis is complete. To archive all system states, click the option labeled **Save all states**, then, supply the directory where the states should be written, and a prefix to use in naming the state files. You can use this prefix to distinguish the states of different runs that might have been written to the same directory. If you do decide to use this option, keep in mind that, depending on the size of your system, archiving states frequently can require significant disk space and delay the process. However, it can be quite helpful if previous analyses have shown interesting behavior at a particular transfer level.

You may also save only the base case for each critical contingency, or save no system states at all, by checking the appropriate option.

---

## PV/QV PV Results

---

To start the PV study, switch to the **PV Results** tab, the last page of the form. From this tab, you can control the progress of the run by initiating, pausing, or aborting it. You can even reset the process to the beginning. This tab also gives you a few different views of the run's output, including the ability to plot various quantities.

To begin the PV analysis, click **Run**. The status field will update with the progress of the analysis. Totals of the Gen MW and Load MW by component for the source and sink will update as these values change during the analysis. To terminate the analysis at any time, click **Stop**.

As the analysis continues, the scenario list on the **Overview** sub-tab will keep you informed of its progress. **Present step size** identifies the current size of the increment the application is using to increase the transfer. **Present Nominal Shift** indicates the size of the transfer that was most recently solved.

The **Save Results** button can be used to save the results to an auxiliary file. The **Load Results** button can be used to load previously saved results from an auxiliary file. The **View detailed results** button opens a text file that contains the detailed results including the values of the tracked quantities at each step for each scenario.

The **Other actions** button provides access to other options available with the PV analysis. The **Save critical contingencies** option will save to an auxiliary file all of the contingencies that were found to be critical for some reason during the analysis. **Restore initial state** will restore the power system to the state when the PV analysis was first started. The remainder of the actions available under **Other actions** are self-explanatory.

The **Plot** sub-tab provides access to the application's data plotting functions. Any of the values designated to [monitor](#) can be plotted using this display. To produce a plot, the **Horizontal axis value** and **Vertical axis value** must be specified, the scenarios to plot must be selected in the **PV Scenario** table by setting the **Plot** field to *YES*, and the elements for which to plot values must be selected under **Plot values for these elements** by setting the **Plot** field to *YES*. An optional **Plot title** can be specified. Once the options are set, click the **Plot** button to produce the plot. The **Clear** button will clear all of the options.

The plot will appear in its own window. You can produce as many plots as the memory on your system will allow. Right-clicking on a plot will expose a local menu with several options. You can send the plot to your printer by clicking **Print**. You can change the printer configuration by selecting **Printer Setup**. You can save the plot as a bitmap, Windows metafile, JPEG, or text file by clicking **Save** and selecting the appropriate file type. Use **Copy** to make a copy of the plot that can be pasted into other applications. Finally, to close the plot window, click **Close**.

Note that the plot tab also has an option for plotting pre-contingency values and for always including the plot of the base case conditions.

The **Track Limits** sub-tab has additional tabs listing the device types that have been selected for tracking. Each of these tabs contains a table of the elements tracked and where that element was located relative to its limits during each step of the analysis using the indicators of *Within Limits*, *Within Range*, *At Max*, and *At Min*.

---

## PV/QV QV Setup

---

The PV curve tool stresses the system by increasing a transfer between injection groups. Eventually, the transfer is increased so much that the power flow can no longer be solved. Under the assumptions of static voltage stability assessment, the point at which the system becomes unsolvable is regarded as the point of voltage collapse. Voltage collapse tends to be a localized phenomenon associated with a lack of reactive voltage support at a bus or group of connected buses. It is important to identify this group of stressed buses so that you can properly direct efforts to reinforce the system. QV analysis is often used to identify the buses that are most prone to a voltage collapse because they are close to having insufficient reactive support.

The design of the PVQV add-on to Simulator acknowledges the important role that QV analysis plays in identifying the buses that are most heavily stressed by a transfer. The approach is straightforward. First, perform a PV curve analysis that models a transfer from source to sink for both base and contingency topologies. This will yield a number of PV curves that terminate at a transfer level that causes the system to become unsolvable for each topology. For each of these critical transfer level / contingency pairs, perform a QV analysis at a set of buses to try to determine which area or areas constitute the epicenter of the collapse. The set of buses at which the QV analysis is performed can be predefined by the user and supplemented by the list of buses that have the lowest voltage magnitudes or the highest VQ sensitivities.

To specify that Simulator should automatically launch a QV analysis after finishing a PV study, check the box labeled **Automatically launch QV analysis at end of PV computation**.

After the PV curve has found all the critical cases it has been asked to identify, it will launch the [QV Curve Tool](#). The buses the QV Curve Tool will analyze will then come from two groups: those that the user has pre-selected, and those that tool automatically identified because they are among the lowest-voltage buses or have the highest VQ sensitivities. To pre-select buses for the QV analysis, switch to the **Buses** sub-tab of the **Quantities to track** tab and toggle the **Draw QV?** field of each bus for which you want to compute a QV curve to **YES**. Whether other buses are automatically selected in light of their voltage or VQ sensitivity is governed by settings stored in the QV options file.

We now continue our look at the QV Setup tab of the PV Curve Tool by describing the use of the remaining controls.

### QV options file

Specify the name of the file that contains settings for a variety of options associated with the QV study in this text box. Use the **Browse** button to locate this file using a file dialog. Among the topics that may be addressed by the QV options file is whether buses should be automatically selected as buses for which to draw QV curves based on their voltage or VQ sensitivity. The easiest way to build a QV options file is to use the [QV Curve Tool](#) to specify the settings of the various options and to save these settings in a file.

### How should scenarios be handled?

QV curves can be drawn for the selected buses either for all the scenarios that the PV tool has processed, or just for the scenarios for which the PV Curve Tool found an unsolvable transfer level. Select the option that describes how you wish the QV Curve Tool to process scenarios from this option group. If you select the **Consider just the unsolved scenarios** option, the QV tool will compute QV curves for the selected buses at the last transfer level at which each scenario was successfully solved. Otherwise, the tool will compute QV curves for the selected buses for each scenario at either the last transfer level at which the scenario was successfully solved or the last transfer level that the PV tool attempted to model.

**QV accelerator settings**

This group of controls enables you to define a voltage window over which each QV curve should be calculated. If you select **Do not use shortcuts; trace the entire QV curve**, each QV curve will be computed over the voltage range specified for the bus in the QV options file. This voltage range may be specific to that bus or may be a window that has been defined for the entire run. If, on the other hand, you select **Trace abbreviated QV curves using these settings...**, the voltage range over which each bus's QV curve will be computed will be customized for each bus according to the values you specify for **Max voltage increase** and **Max voltage decrease**. For example, if a bus is currently at 0.9 per-unit voltage for a particular scenario and you specify the max voltage increase and decrease to be 0.1 and 0.3, respectively, then the QV curve for that bus for that scenario will be traced over the voltage range 0.6 pu to 1.0 pu.



---

**PV/QV QV Curves**

---

Another voltage stability function currently included is the ability to compute QV curves for any bus in the system. Simply select **QV Curves** on the **PVQV** ribbon group on the [Add Ons](#) ribbon tab. The QV Curve dialog allows you to specify the buses to be monitored, set defaults for the QV curves, and run the QV analysis. These three topics are handled on the following pages of the QV Curve dialog:

[Buses](#)

[Quantities to Track](#)

[Options](#)

[Results](#)

When all options have been set, click **Run** to start the QV analysis. The progress of the analysis can be monitored from the **Results** tab.

When you are finished, simply click **Close**. The results will remain in memory as long as the case remains in memory. Please note that QV curve results and option settings are not saved with the case.

---

**PV/QV QV Curve Buses**

---

To designate the buses for which you would like to calculate QV curves, toggle the **Selected?** field to *YES*. If you would like to specify specific solution parameters (min volt, max volt, and step size) for any bus, simply type the numbers in the appropriate cells. If you leave any cells blank, default values will be used for those parameters. Only non-default values will be shown in these cells. If you change a value in these cells to a default setting, the cell will be automatically set blank.

Alternatively, you may type a range of bus numbers to add them to the list of buses that will have a QV curve calculated, or you may type the name of a bus. Furthermore, you may load a list of buses, as well as other option settings, from an [auxiliary data file](#). To create such a file to store your option settings, press the **Save** button that appears near the bottom of the display.

To load settings you saved previously in an [auxiliary data file](#), click the **Load** button. You can search for the file instead of typing its name. Once the name of the file appears in the text field, press **Open** to import the settings stored in the file.

You also can check the box **Additionally, automatically draw curves for...** and then the number of **lowest-voltage buses** and **highest dv/dq buses** that you require. These buses will be selected from the limit group indicated in the field **from limit group**.

---

## PV/QV QV Curve Options

---

The **Options** page of the QV Curve dialog contains three tabs labeled **Solution**, **Output** and **Contingencies**.

### Solution

In the three text fields enclosed in the **Default solution parameters** group box, specify the values of voltage step size, minimum voltage, and maximum voltage to use when computing QV curves for buses that do not have specific values set for these parameters.

If you do not understand the meaning of these parameters, consider how a QV curve is computed: a fictitious generator (i.e. voltage source) is placed at the monitored bus. Its set point voltage is varied in steps of the specified size between the specified maximum and minimum voltages, and the Mvar injection at the bus is calculated and recorded at each step.

To make the changes that you have made register with the QV curve calculation, click the **Set Options** button. To restore the factory-default settings, click **Restore Defaults**.

To set power flow solution parameters, click the **Global solution options** button. Be aware that the changes you make in the resulting dialog affect all power flow computations in Simulator.

### Contingencies

QV curves can be calculated for the specified buses for both base and contingency conditions. To analyze a set of contingencies, you must define the contingency set using the separate [Contingency Analysis Dialog](#), which you can access from run mode by selecting **Contingency Analysis** from **Tools** ribbon tab. In order to have the QV curves drawn automatically for a contingency, the **QV Autoplot** field belonging to the contingency must be set to **YES**.

To have the QV curve calculation analyze these contingencies for each bus, check the box labeled **Process each of the currently defined contingencies**. Check the box labeled **Skip base case** for avoiding the computation of QV curves for base conditions.

### Output

The **Output** tab allows you to set a location for saving the results of the QV analysis to a file. To save the QV analysis to a text file, you must first check the box labeled **Save results in file**. Once this box is checked, you can then either type in a directory location for saving the data, or click the **Browse** button and select a file to save as. You can also customize the file prefix and extension to be used when Simulator names the output files.

Another option on the **Output** tab is the check box labeled **Plot curves as they are computed**. When this option is checked, the QV curve displays will be updated and drawn during each step of the QV analysis process. In order to have the QV curves updated and drawn for a contingency during each step of the analysis, the **QV Autoplot** field belonging to that contingency must be set to **YES** on the [Contingency Analysis Dialog](#).

When plotting the QV curves, there are two options for how to treat the Mvar portion of the curve. Q can either represent **The output of the fictitious synchronous condenser** or **The total reactive injection at the bus, including shunts but excluding load**.

---

**PV/QV QV Curve Results**

---

To perform the QV curve calculations, switch to the **Results** tab and press the **Run** button. The QV curve calculation will respond by calculating QV curves for each of the specified scenarios (i.e. for each bus / contingency pair). When it finishes the QV curve calculation for a scenario, it will record the critical points for the curve in a [case information display](#) found on the **Listing** sub-tab. These critical points include (V0, QV0), (Vmin, QVmin), (Vmax, QVmax), and (VQmin, Qmin).

Each scenario can be identified using the bus number and name fields together with the case name field, which simply specifies either *BASECASE* or the name of the contingency. As with all case information displays, the QV Results case information display has a local menu that offers a number of options. From the local menu, you can plot the QV curve for a scenario, record the (Q, V) pairs that comprise each curve to a text file or to an Excel spreadsheet, and clear the results.

The **Plots** sub-tab allows plotting any values that have been set with [Quantities to Track](#). The plots can be created in the same manner as described for the **Plot** sub-tab of the [PV Results](#).

The information contained in the **Track Limits** sub-tab contains the same type of information as described in the **Track Limits** sub-tab of the [PV Results](#).

The QV curve calculation interface remains active while the curves are calculated. You can terminate the run at any time by clicking **Stop**.

---

**PV/QV Refine Model**

---

Simulator PVQV has the ability to refine the system model to fix modeling idiosyncrasies that cause premature loss of convergence during the PV and QV curve studies. This option is available on the [Add Ons](#) ribbon tab in the **PVQV** ribbon group.

The user can refine the case in the following ways:

**Fix transformer taps**

If there are transformers that have Vmax and Vmin that are very close together, the power flow may have a difficult time converging. This option allows the user to fix all transformer taps at their present values if their Vmax - Vmin is less than or equal to the user specified tolerance.

**Fix shunts**

If there are shunts that have Vmax and Vmin that are very close together, the power flow may have a difficult time converging. This option allows the user to fix all shunts at their present values if their Vmax-Vmin is less than or equal to the user specified tolerance.

**Take units off AVR control**

If there are generators that have Qmax and Qmin that are very close together, the power flow may have a difficult time converging. This option allows the user to remove these units from AVR control, thus locking their MVAR output at its present value, if their Qmax - Qmin is less than or equal to the user specified tolerance.

These refinements will only be applied to those areas or zones that have the **Apply?** field set as *YES* in this dialog box. This field can be changed by right-clicking on it and selecting Toggle all to *YES/NO* or by double-clicking on the field.

---

## PowerWorld Simulator Optimal Power Flow Overview

---

**Note:** The OPF option in PowerWorld Simulator is only available if you have purchased the OPF add-on to the base package. To learn more about the OPF, please feel free to read through the information contained in these help files. [Contact PowerWorld Corporation](#) for details about ordering the OPF version of Simulator.

The PowerWorld Simulator (Simulator) is an interactive power system simulation package designed to simulate high voltage power system operation. In the standard mode Simulator solves the power flow equations using a Newton-Raphson power flow algorithm. However with the optimal power flow (OPF) enhancement, Simulator OPF can also solve these equations using an OPF. In particular, Simulator OPF uses a linear programming (LP) OPF implementation.

All of the OPF commands and options are accessed using the LP OPF main menu item. Other commands in this menu are used to specify input options, see results, and store/retrieve OPF specific data into auxiliary files.

The purpose of an OPF is to minimize an [objective \(or cost\) function](#) by changing different [system controls](#) taking into account both [equality and inequality constraints](#) which are used to model the power balance constraints and various operating limits.

In Simulator OPF the LP OPF determines the optimal solution by iterating between solving a standard power and then solving a linear program to change the system controls to remove any limit violations. See [OPF Primal LP](#) for more details.

---

## OPF Objective Function

---

The objective of the OPF algorithm is to minimize the OPF objective function, subject to various equality and inequality constraints. Since the objective of the OPF is to minimize an objective function, what objective function is used has a significant impact on the final solution.

Currently two objective functions are available in Simulator OPF: Minimum Cost and Minimum Control Change. Minimum Cost attempts to minimize the sum of the total generation costs in specified areas or super areas. Minimum Control Change attempts to minimize the sum of the absolute values of the changes in the generation in the specified areas or super areas.

To include an area or super area in the OPF objective function, simply change the Area AGC Status field to "OPF" on the [OPF Area Records Display](#) or the Super Area AGC Status field to "OPF" on [the OPF Super Area Records Display](#). This gives you great flexibility in defining the OPF study. For example you can set the OPF to minimize costs for the entire system, or just selected areas or super areas.

---

## OPF Equality and Inequality Constraints

---

In solving a constrained optimization problem, such as the OPF, there are two general classes of constraints, equality and inequality. Equality constraints are constraints that always have to be enforced. That is, they are always "binding". For example in the OPF the real and reactive power balance equations at system buses must always be satisfied (at least to within a user specified tolerance); likewise the area MW interchange constraints. In contrast, inequality constraints may or may not be binding. For example, a line MVA flow may or may not be at its limit, or a generator real power output may or may not be at its maximum limit.

An important point to note is because the OPF is solved by iterating between a power flow solution and an LP solution, some of the constraints are enforced during the power flow solution and some constraints are enforced during the LP solution. The constraints enforced during the power flow are, for the most part, the constraints that are enforced during any power flow solution. These include the bus power balance equations, the generator voltage set point constraints, and the reactive power limits on the generators. What differentiate the LP OPF from a standard power flow are the constraints that are explicitly enforced by the LP. These include the following constraints:

[Equality Constraints](#)

[Inequality Constraints](#)



---

## OPF Equality Constraints

---

### Area MW Interchange

The area MW interchange constraints are enforced during the LP for those areas that have an **AGC Status** equal to "OPF" provided the area is not part of a super area that is also set on AGC. The **AGC Status** field for an area can be set using the [OPF Area Records display](#), while the **AGC Status** field for the super area (if any) is set using the [OPF Super Area Records display](#). Areas whose interchange is enforced during the LP do not have their interchange enforced during the power flow solution; during the power flow these areas are treated as though they were off of AGC (and hence the output of generators in that area is not varied during the power flow).

It is perfectly acceptable to have some areas on "OPF" AGC control and to have other areas on the more traditional power flow area AGC such as "ED" or "Part. AGC." The interchange for such areas is controlled during the power flow solution.

Following a successful solution, marginal costs are calculated for the area interchange constraints; these values are displayed on the [OPF Area Records display](#) and can be contoured. See [OPF Marginal Costs](#) for details.

### Bus MW and Mvar power balance

Enforced during the power flow solution. Following a successful solution, marginal costs are calculated for the bus MW (real power) balance constraint; these values are displayed on the [OPF Bus Records display](#) and can be contoured.

### Generator Voltage Setpoint

Enforced during the power flow solution. Following a successful solution, marginal costs are calculated for the voltage setpoint constraint; these values are displayed on the [OPF Bus Records display](#).

### Super Area MW Interchange

Super area interchange constraints are enforced similar to the area constraints. That is, super area interchange constraints are enforced during the LP only for those super areas that have an **AGC Status** equal to "OPF." The **AGC Status** field can be set using the [OPF Super Area Records display](#). During the power flow solution such super areas are treated as though they were off of AGC.

### Interface MW limits when treated as Equality

Interface MW limits are enforced during the LP solution. Interface MW limits are normally treated as inequality constraints (see [Inequality Constraints](#)), however they can optionally be treated as equality constraints. See the [Interface Dialog](#) for information on how to treat the limit as an equality constraint.

### Transmission Line and Transformer (Branch) MVA limits

Branch MVA limits are enforced during the LP solution. Branch MVA limits are normally treated as inequality constraints (see [Inequality Constraints](#)), however they can optionally be treated as equality constraints. See the [Line Transformer Dialog](#) for information on how to treat the limit as an equality constraint.

---

## OPF Inequality Constraints

---

The following classes of inequality constraints are enforced during the OPF solution.

### Generator real power limits

Generator real power limits are enforced during the LP solution.

### Generator reactive power limits

Generator reactive power limits are enforced during the power flow solution.

### Interface MW limits

Interface MW limits are enforced during the LP solution. In short, interface records are used to represent the aggregate flow through a number of different devices (see [Interface Records](#) for details). During the LP the MW flow through the interface is constrained to be less than or equal to a user specified percentage of its limit, provided the interface is active for enforcement. For an interface to be active for enforcement the following three conditions must be met:

- Interface enforcement must not be disabled for the case. This field can be set from either the [OPF Options and Results dialog](#) or the [OPF Interface Records display](#). The default is that case interface enforcement is not disabled. Also note that interface flow is limited to a percent of its limit as specified by the interface's [Limit Monitoring Settings](#).
- Interface enforcement must be active for at least one of the interface's areas. Note, an interface is assumed to be in each area that contains at least one of its components. This field can be set from [the OPF Area Records display](#). Note: the default is that interface enforcement is **not active**, so be sure to activate this if you want these constraints enforced.
- Enforcement must be active for each individual interface. This field can be set from [the OPF Interface Records display](#) or in the [Limit Monitoring Settings Dialog](#). The default is active.

Each interface that is active for enforcement is modeled as an inequality constraint, which may be either binding or not binding. If the constraint is not binding then it does not impact the solution. If a constraint is binding then it has an associated marginal cost of enforcement, which is shown on the [OPF Interface Records display](#).

### Transmission Line and Transformer (Branch) MVA Limits

Transmission line and transformer (branch) MVA limits are enforced during the LP solution. During the LP the branch line flow is constrained to be less than or equal to a user specified percentage of its limit, provided the branch is active for enforcement. For a branch to be active for enforcement the following three conditions must be met:

- Line/Transformer enforcement must not be disabled for the case. This field can be set from either the [OPF and Results Options dialog](#) or the [OPF Line/Transformer Records display](#). The default is that case line/transformer enforcement is not disabled. Also note that the branch flow is limited to a percent of its limit as specified by the branch's [Limit Monitoring Settings](#).
- Branch enforcement must be active for the branch's area. For tie-lines enforcement must be active for either area. This field can be set from the [OPF Line/Transformer Records display](#). The default is that branch enforcement is **not active**, so be sure to activate this if you want these constraints enforced.
- Enforcement must be active for each individual branch. This field can be set from the [OPF Line/Transformer Records display](#) or in the [Limit Monitoring Settings Dialog](#). The default is active.

Each branch that is active for enforcement is modeled as an inequality constraint, which may be either binding or not binding. If the constraint is not binding then it does not impact the solution. If a constraint is binding then it has an associated marginal cost of enforcement, which is shown on the [OPF Line/Transformer Records display](#).

---

## Determining Set of Active Inequality Constraints

---

A key issue in quickly solving the OPF is for the LP to effectively determine the set of active inequality constraints. Currently this includes the line MVA limits and the interface MW limits. Because the speed of the LP varies as the cube of the number of constraints active in the LP basis, it is extremely important to keep this number as small as possible. Therefore it would be very computationally prohibitive to setup an inequality constraint for each transmission line and interface (except in very small systems.)

The solution of setting up constraints **only** for those inequality constraints that are actually violating their limits is a step in the right direction, but suffers from the problem that during a solution a line may initially be violating its limit and then after the first iteration it is no longer violating. However if it is not subsequently included as a constraint during the next iteration the solution may simply oscillate between enforcing/unenforcing this constraint. This problem can be resolved by keeping that constraint in the basis even though it is no longer binding.

However this raises a question about how to handle these constraints during future OPF solutions. For example what would happen if a user solved the OPF, and then immediately resolved the OPF. Following the first solution the constraint would be enforced so that it may actually be less than its limit. However if this constraint is not included in the LP basis during the next solution the constraint may immediately violate during the first iteration, requiring a number of iterations just to return to the original initial solution.

Simulator solves this issue by keeping track of the enforced constraints from one solution to the next. Constraints are only removed from the basis if they fall below a specified percentage of their limit. This percentage is enterable on the [Constraint Options page](#) of the [OPF Options and Results Dialog](#). This prevents the set of constraints in the basis from building up over time as a number of different system conditions (and hence constraints) are studied. Also, at any time this set of constraints can be cleared using the **Initialize OPF Button** on the [OPF Options and Results Dialog](#).

Also, the user is free to specify that a particular constraint **always** be included in the basis. This is done by toggling the **Constraint** field to "Always" on the [OPF Line/Transformer Records](#) or [OPF Interface Records](#) displays.

---

## OPF Unenforceable Constraints

---

The goal of the LP OPF is to minimize the objective function subject to the user specified constraints. However there is no guarantee that it is even possible to simultaneously satisfy all of the specified constraints. In fact, it is quite easy to create a system in which all of the constraints **cannot** be enforced. A simple example is a two bus system consisting of a single generator supplying a single load through a transmission line. If the transmission line MVA rating is below the MVA of the load then it is impossible to supply this load while simultaneously satisfying the transmission line constraint. In Simulator OPF such a situation is known as an unenforceable constraint. In studying large systems, such as the U.S FERC 715 cases, such situations actually appear to be quite common. Seemingly unenforceable constraints are often due to a lack of controls available to the LP OPF or due to faulty limits entered in the case. In such cases unenforceable constraints can be corrected by making more controls available to the LP OPF or correcting the limits.

Simulator OPF allows you to solve systems with unenforceable constraints by only enforcing those constraints that have a marginal cost below a user specified tolerance. These tolerances are specified on the [OPF Options Constraint Options Page](#). Any constraints that have marginal costs above these values are not enforced, including any unenforceable constraints. This functionality is implemented in Simulator OPF through the use of slack variables. Slack variables are artificial variables introduced during the LP solution in order to satisfy the constraints with the slack variable costs equal to the user specified values. Then, during the LP solution the slack variables are usually removed from the LP basis. The only time this does not occur is if the constraint can not be enforced with a marginal cost less than the specified value. The number of unenforceable constraints are shown on the [OPF Option Solution Results Page](#).

---

## OPF Marginal Costs

---

During any constrained minimization there is a cost associated with enforcing the equality constraints and the binding inequality constraints. These costs are known as the marginal costs.

In Simulator OPF marginal costs are calculated for the following record types:

### Bus MW Equality Constraints

The Bus MW marginal costs tell the incremental cost to supply one additional MW of load at the specified bus. These values can be viewed on the [OPF Bus Records display](#); they can also be contoured or viewed on the one-lines using [bus fields](#).

In the absence of any binding inequality constraints (such as Line MVA constraints) all of the bus marginal costs in an area should be identical. Bus marginal costs can only be determined for buses that are in areas or super areas on OPF control.

### Area MW Equality Constraints

The Area MW marginal costs tell the incremental cost for the specified area to import one additional MW of load **from the system slack bus**. These values can be viewed on the [the OPF Area Records display](#); they can also be contoured or viewed on the one-lines [using area fields](#). In the absence of any binding inequality constraints the area MW marginal cost is identical to the bus MW marginal costs for all the buses in the area. When there are binding inequality constraints this is no longer the case.

### Super Area MW Equality Constraints

The Super Area MW marginal costs are identical to the area marginal costs except they apply to super areas rather than areas.

### Interface MW Constraints

The Interface MW marginal costs tell the incremental cost of enforcing the interface MW constraints. These values are only nonzero if the interface constraint is actually active (binding); they can be viewed using the [OPF Interface Records display](#).

### Line/Transformer MVA Constraints

The Line/Transformer marginal costs tell the incremental cost of enforcing the line or transformer MVA constraint. These values are only nonzero if the line or transformer constraint is actually active; they can be viewed using the [OPF Line/Transformer Records display](#).

---

**OPF Primal LP**

---

Go to the [Add Ons](#) ribbon tab and select **Primal LP** from the **OPF** ribbon group to solve the OPF using the primal LP algorithm.

In Simulator OPF the LP OPF determines the optimal solution by iterating between solving a standard power flow and then solving a linear program to change the system controls to remove any limit violations. The basic steps in the LP OPF algorithm are

1. Solve the power flow
2. Linearize the power system about the current power flow solution. Both constraints and controls are linearized.
3. Solve the linearly-constrained OPF problem using a primal LP algorithm, computing the incremental change in the control variables. Slack variables are introduced to make the problem initially feasible. That is, the slack variables are used to satisfy the equality and inequality constraints. The slack variables typically have high costs so that during the iteration the slack variables change to satisfy the constraints. The LP then determines the optimal, feasible solution for the linear problem.
4. Update the control variables and resolve the power flow.
5. If the changes in the control variables are below a tolerance then the solution has been reached; otherwise go to step 2.
6. Finish by resolving the power flow.

---

## OPF Future Enhancements

---

While we certainly plan on introducing additional functionality in future releases, we do want to be as clear as possible about what functionality is not currently provided.

The current version of Simulator OPF allows users to calculate the optimal solution to a power system using generator real power MW outputs and phase shifters as controls, while enforcing area, super area, interface MW and line/transformer MVA constraints. Marginal losses can also be included in the OPF calculation.

Some functionality that is **not** included in the current version of Simulator OPF, and which we hope to include in future versions, include the following:

- Enforcing bus low/high voltage magnitudes as limits
- Including additional devices as controls, such as generator voltage setpoints, LTC transformers, switched shunts.
- Allowing the optimization of different cost functions.
- Additional functionality as suggested by customers.



---

## OPF Options and Results

---

The OPF Options and Results dialog allows you to customize the OPF solution. To display this dialog, go to the [Add Ons](#) ribbon tab and select **Options and Results** from the **OPF** ribbon group. The dialog consists of three general pages; Options, Results and LP Solution Details.

The Options page has four tabs as well, [Common Options](#), [Constraint Options](#), [Control Options](#), and [Advanced Options](#)

The Results page has four tabs, [Bus MW Marginal Price Details](#), [Bus Mvar Marginal Price Details](#), [Bus Marginal Controls](#), and [Solution Summary](#).

The LP Solutions Page has five tabs, [All LP Variables](#), [LP Basic Variables](#), [LP Basis Matrix](#), [Inverse of LP Basis](#), and Trace Solution.

The dialog also has several buttons at the bottom of the display:

### **OK, Cancel**

Select to close the dialog. Selecting **OK** saves your changes while **Cancel** does not. Note that changes are also saved anytime you select **Solve LP OPF** or **Single Outer Loop**.

### **Solve LP OPF**

Solves the OPF using the Primal LP algorithm. Equivalent to selecting [Primal LP](#) from the **Optimal Power Flow** ribbon group on the [Add Ons](#) ribbon tab.

### **Single Outer Loop**

Does a single outer loop of the Primal LP algorithm.

### **Initialize LP OPF**

Returns the LP OPF variables to their original states.

### **Print**

Prints the selected page of the dialog.

### **Help**

Displays this help page. To view help for a particular page place the cursor on the page and press the F1 key.

---

**OPF Options: Common Options**


---

The [OPF Dialog](#), Common Options page displays general options associated with the OPF solution. The display contains the following fields:

**[Objective Function](#)**

Allows a choice of solving the LP using either a minimum cost or a minimum control change objective function.

**LP Control Variables**

**Disable All Phase Shifter Controls**

Prevents phase shifters from attempting to control devices during the OPF solution.

**Disable All Generator MW Controls**

Prevents generators from shifting MW output during the OPF solution.

**Disable All Load MW Controls**

Prevents loads from shifting MW demand during the OPF solution.

**Disable Area-toArea MW Transaction Controls**

Prevents MW transactions between areas from being dispatched during the OPF solution.

**Disable DC Transmission Line MW Controls**

Prevents shifting of DC Transmission Line MW transfers during the OPF solution.

**LP Options**

**Maximum Number of LP Iterations**

Maximum number of allowable iterations for the LP portion of the LP OPF. How many iterations are required to obtain a solution depends, among other things, upon the number of breakpoints in the control cost models. Since each LP iteration can only move from one breakpoint to the next, the finer the model the more iterations required. However the LP is quite fast so a large number of iterations can be performed quite quickly. Default = 1000.

**Phase Shifter Cost (\$ / Degree)**

Specifies the assumed cost for moving phase shifting transformer taps away from their initial values. The purpose for this fictitious cost is approximate the cost of actually changing the angle of a phase shifting transformer, and to avoid large changes in phase shifter angles that have very little impact on the system. This field may be zero. Default = \$ 0.10 / Degree.

**Calculate Bus Marginal Cost of Reactive Power**

When this option is checked, the OPF algorithm will also calculate the marginal cost of reactive power at each bus. Typically the result of interest from the OPF algorithm is the MW marginal cost of each bus (the LMP), but the MVAR marginal cost can be determined as well.

**Save Full OPF Results in PWB File**

When checked, Simulator will store the full set of results, including the LP matrix, in the PowerWorld Binary case file.

**Do Detailed LP Logging**

When checked, Simulator will write details on the LP algorithm solution during each pivot of the LP matrix. This is useful for debugging LP solution issues when running a LP OPF solution.

**Start with Last Valid OPF Solution**

When checked, Simulator will start the LP OPF solution process using the most recent OPF solution as the initial conditions.

---

**OPF Options: Constraint Options**


---

The OPF Dialog, Constraint Options page displays options associated with the enforcement of the constraints by the OPF. The display contains the following fields:

### **Line/Transformer Constraints**

#### **Disable Line/Transformer MVA Limit Enforcement**

Select to disable enforcement of Line/Transformer MVA constraints for the entire case.

#### **Percent Correction Tolerance**

Specifies a tolerance for the enforcement of line/transformer MVA flows. The tolerance is necessary to prevent solution oscillations due to the non-linear nature of the actual constraints.

Violated elements are always enforced to their limits multiplied by the MVA Enforcement Percentage. If power systems were completely linear then following the LP solution the constraint would actually be equal to this value. However because of nonlinearities, the constraint is close to this value but usually not identical to the value. The Percent Correction Tolerance is used to tell the OPF how close is close enough. Provided all the constraints are violating their limits by less than the correction tolerance percentage the optimal solution is assumed to have been found. You may set this value as low as you like, but setting it too close to zero may result in convergence difficulties. The default is 2 percent.

#### **MVA Auto Release Percentage**

Specifies a MVA level at which transmission lines can be released as an OPF constraint equation if the branch MVA flow falls below the level specified.

#### **Maximum Violation Cost (\$/MWhr)**

If a branch MVA limit cannot be enforced during an OPF solution, the branch will be assigned a fictitious cost of enforcement equal to this value. This value is usually rather large in order to easily determine where the unenforceable constraint is occurring. The default value is 1000 \$/MWhr.

#### **Enforce Line/Transformer MW Flow Limits (Not MVA)**

Checking this box will cause Simulator to treat the limits of the transmission elements as MW limits instead of MVA limits. Thus Simulator will report violations on these elements in the OPF based on the MW flow of the element versus the element's MVA rating.

### **Interface Constraints**

#### **Disable Interface MW Limit Enforcement**

Select to disable enforcement of Interface MW constraints for the entire case.

#### **Percent Correction Tolerance, MW Auto Release Percentage, Maximum Violation Cost (\$/MWhr)**

These fields are equivalent to the entries described above for Line/Transformer MVA Constraints except that they apply to Interface MW constraints.

#### **Monitor/Enforce Contingent Interface Limits**

This option allows you to choose if contingency elements in interfaces should be enforced during the OPF solution. Even if they are not enforced, the flows on the remaining elements in the interface will be monitored. The choices you have for enforcing contingency elements

in interfaces during the OPF are Never, Power Flow/OPF but not CA/SCOPF, or All Applications including CA/SCOPF. The reason CA ([contingency analysis](#)) and SCOPF ([Security Constrained OPF](#)) are singled out is because those two tools are already looking at contingency actions separately, outside of the interface definitions. Therefore it may be desired to ignore the inclusion of contingency elements within an interface definition when using these two tools.

### **Phase Shifting Transformer Regulation Limits**

#### **Disable Phase Shifter Regulation Limit Enforcement**

Select to disable enforcement of Phase Shifter regulation limit constraints for the entire case.

For a phase shifter, there are two limits which can be enforced during the OPF—the MVA branch rating, and the regulation minimum and maximum MW flow. Checking this option will globally disable the enforcement of the regulation minimum and maximum MW flow for all phase shifters.

#### **In Range Cost (\$/degreehr)**

The cost of changing a phase shifter angle away from zero degrees in \$/degreehr, when the angle is within the allowable range of the phase shifter.

#### **Maximum Violation Cost (\$/MWhr)**

If a phase shifter regulation limit cannot be enforced during an OPF solution, it will be assigned a fictitious cost of enforcement equal to this value. This value is usually rather large in order to easily determine where the unenforceable constraint is occurring. The default value is 1000 \$/MWhr.

#### **Limit Monitoring Settings...**

This button opens the [Limit Monitoring Settings](#) dialog, which allows you to change the enforcement percentages for monitored elements.

---

**OPF Options: Control Options**


---

The LP OPF Dialog, Control Options page displays options for generator control and power flow solution. The display contains the following options:

### **Generator Control Options**

#### **Allow Commitment of Fast Start Generators**

If this option is checked, then generators designated as Fast Start generators can be turned on or "committed" if the OPF routine determines that doing so would reduce the overall generating costs of the system.

See the Fast Start description in the help on [OPF Generator Records](#) for a more detailed description of the Fast Start option of generators.

#### **Allow Decommitment of Fast Start Generators**

If this option is checked, then generators designated as Fast Start generators can be turned off or "de-committed" if the OPF routine determines that doing so would reduce the overall generating costs of the system.

See the Fast Start description in the help on [OPF Generator Records](#) for a more detailed description of the Fast Start option of generators.

### **Modeling Generators without Piecewise Linear Cost Curves**

The following fields specify how the OPF should handle generators that are specified as having a cubic cost model. Because the OPF is based upon an LP implementation, all control costs must be modeled using piecewise linear cost curves. These options permit an automatic conversion of cubic models to piecewise linear models. Alternatively, you can very easily convert the cubic models manually using the **# Cost Curve Points** field on [the OPF Generator Records display](#) or using the [generator dialog](#).

#### **Generators Cost Models**

This field specifies how generators with cubic cost models should be handled in the OPF. The field has three values

**Ignore Them** - Generators with cubic cost models are Ignored during the OPF solution. That is, they are considered as though their AGC status was off.

**Change to Specified Points per Curve** - A piecewise linear cost model is automatically inserted for the generator with a fixed number of points specified in the **Total Points Per Cost Curve** field described below. This curve will approximate the generator's cubic cost model as closely as possible; the existing cubic model is not modified. This is the default value.

**Change to Specified MWs per Segment** - A piecewise linear cost model is automatically inserted for the generator such that each segment in the cost model covers the amount of MWs specified in the **MWs per Cost Curve Segment** field described below. This curve will approximate the generator's cubic cost model as closely as possible; the existing cubic model is not modified.

#### **Total Points Per Cost Curve**

Specifies the total number of segments that should be automatically inserted into the piecewise linear cost models for those generators that are modeled using cubic cost functions. This is only done if the **Generator Cost Modeling** field is **Change to Specified Points per Curve**. Default = 5.

### MWs per Cost Curve Segment

Specifies the number of MWs for each segment of the piecewise linear cost models that are automatically inserted for those generators that are modeled using cubic cost functions. This is only done if the **Generator Cost Modeling** field is **Change to Specified MWs per Segment**. Default = 10 MW.

### Save Existing Piecewise Linear Cost Curves

Generators that are modeled with cubic cost curves may have existing piecewise linear cost curves which may have been manually entered by the user. These curves may or may not resemble the cubic cost function. During the OPF solution the existing piecewise linear cost curves are replaced with the auto-created cost curves. If this option is checked then the existing piecewise linear cost curves are restored at the end of the OPF. The default and recommended option is false since this allows one to view the actual cost curves used by the OPF.

If you would like to use a particular piecewise linear cost function simply make sure that the generator is modeled using the piecewise linear model, which can be set on the [OPF Generator Records](#) display.

## Modeling of OPF Areas/Superareas

### During the Initial OPF Power Flow Solution

Choose what manner of generation control you wish to be employed in the FIRST power flow solution the OPF will perform, which will establish the base case load flow condition for performing the subsequent OPF generation dispatch.

### During Stand-Alone Power Flow Solutions

Choose what manner of generation control you wish to be employed in all load flow solutions FOLLOWING the initial load flow solution. In other words, after the LP OPF routine has determined the new generation dispatch, what type generation dispatch should be used during the normal load flow solution.

NOTE: it is NOT recommended that you use Economic Dispatch in this case, although it is an available option. The reason it is not recommended is that you will remove the optimal dispatch (including constraints) just determined by the OPF in favor of lowest cost economic dispatch, which will likely result in the re-introduction of overloaded elements that were corrected by the OPF dispatch in the first place.

### Treat Area/Superarea MW Constraints as unenforceable even when the ACE is less than the AGC Tolerance (default is checked)

This option by default is checked, which means that if Simulator solves an OPF and the generators in an area or super area are all either at their minimum or maximum outputs, and the area control error is still not 0, then the area MW constraint is considered unenforceable and Simulator will assign a large penalty cost (usually \$5,000/MWhr) to the LMP's of the area. However, if you deselect this option, you give Simulator the additional control of checking the ACE mismatch against the AGC tolerance for the area, and if the ACE mismatch is less than the AGC tolerance, Simulator will "acquire" the mismatch amount from the system slack bus for the area, and the area will not be considered unenforceable in the OPF solution. The AGC tolerance is usually small, and therefore the amount of power taken from the system slack bus will also be small. The purpose of this option is to allow for valid LMP solutions in an area that is meant to reach its full minimum or maximum generator capability, but that losses and numerical rounding results in the ACE mismatch deviating

slightly from 0. It is not advised to uncheck this option if you have areas with large AGC tolerances.



---

**OPF Options: Advanced Options**

---

**Detection of LP Cycling**

This option allows for advanced tweaking of when cycling is detected in the LP. It is recommended that users leave the default settings in place.

**Minimum Number of Degenerate Iterations to Assume Cycling**

To prevent inaccurate identification of cycling in the LP solution, the minimum number of degenerate iterations before cycling is identified can be specified manually in this box.

**Minimum Number of Degenerate Iterations Multiplied by Number of Tableau Rows**

To prevent inaccurate identification of cycling in the LP solution, the minimum number of degenerate iterations as a multiple of the number of tableau rows can be specified manually in this box.

**Number of Sequential Degenerate Iterations for Last Case**

This displays the number of degenerate iterations detected when performing the most recent OPF solution.

**Power Flow Recalculation**

Choose one of the options to determine how often the power flow is resolved. The three options are, "When total generator MW change > than tolerance"; "After each LP solution"; and "Only at end of LP OPF". If the power flow is not resolved, linear sensitivities are used to determine which constraints enter the LP tableau.

**Total Generator Change Tolerance (MW)**

Specifies the total generator change tolerance. The default is 500 MW.

**After each LP solution**

Resolves the power flow upon completion of each LP solution

**Only at end of LP OPF**

Resolves the power flow only after the LP OPF has completed.

---

**OPF Options: Solution Summary**


---

The OPF Dialog, Solution Summary page displays general results from the last OPF solution. Click the **Save as Aux** button to save these results to a Simulator Auxiliary file.

The display contains the following fields, none of which can be directly changed:

### General Results

#### **Solution Start Time, Solution End Time**

The starting time and ending time of the OPF solution algorithm.

#### **Total Solution Time, Last Solution Status**

Time and status of the last OPF solution.

#### **Number of LP Iterations**

Total number of LP iterations used during the last OPF solution. The maximum number of iterations is specified in the [Maximum Number of Iterations](#) field of the General Options page.

#### **Initial Cost Function Value**

Initial value of the [OPF cost function](#). During the OPF the solution algorithm seeks to minimize the cost function, subject to the equality and inequality constraints.

#### **Final Cost Function Value**

Final value of the OPF cost function.

#### **Final Slack Cost Value**

The slack cost value is an artificial cost that is only non-zero when there are one or more unenforceable constraints.

#### **Final Total Cost Value**

The addition of the final cost function value and the final slack cost value.

#### **Number of Buses in OPF**

This field contains the total number of buses that are in areas or super areas that are on OPF control. Thus this field need not be equal to the total number of buses in the case. Marginal costs are only calculated for buses in OPF controlled areas or super areas.

#### **Highest Bus Marginal Cost, Lowest Bus Marginal Cost, Average Bus Marginal Cost**

Highest, lowest and average marginal cost for the buses that are in OPF controlled areas or super areas.

#### **Bus MC Standard Deviation**

The standard deviation of the Bus Marginal Cost.

### Area and Superarea Constraints

#### **Unenforceable Area Constraints**

Number of unenforceable area constraints.

#### **Unenforceable Superarea Constraints**

Number of unenforceable superarea constraints.

### Line MVA Constraints

The **Line MVA Constraints** fields present results associated with the enforcement of the line MVA constraints.

#### Number of Initial Violations, MVA Sum of Initial Violations

Total number of lines that initially exceeded their MVA limits and were eligible for enforcement by the OPF. For these lines only, the **MVA Sum of Initial Violations** field contains sum of the absolute values of the line's actual MVA flow minus the line's MVA limit.

#### Number of Binding Lines

Total number of lines that are constrained to their limit value.

#### Highest Line MVA Marginal Cost

The highest Marginal Cost for an MVA change on a line.

#### Number of Unenforceable Violations

Total number of lines whose MVA flows can not be enforced by the OPF using the available controls.

#### MVA Sum of Unenforceable Violations

For all the unenforceable lines, this field contains the sum of the absolute values of the line's actual MVA flow minus the line's MVA limit.

### Interface MW Constraints

The **Interface MW Constraints** field present results associated with the enforcement of the interface MW constraints.

#### Number of Initial Violations, MW Sum of Initial Violations

Total number of interfaces that initially exceeded their MW limits and were eligible for enforcement by the OPF. For these interfaces only, the **MW Sum of Initial Violations** field contains the sum of the absolute values of the interface's actual MW flow minus the interface's MW limit.

#### Number of Binding Interfaces

Total number of interfaces that are constrained to their limit value.

#### Highest Interface MW Marginal Cost

The highest Marginal Cost for an MVA change on an interface.

#### Number of Unenforceable Violations

Total number of interfaces whose MW flows can not be enforced by the OPF using the available controls.

#### MW Sum of Unenforceable Violations

For all the unenforceable interfaces, this field contains the sum of the absolute values of the interface's actual MW flow minus the interface's MW limit.

### Generator MW Control Limit Violations

The **Generator MW Control Limit Violations** fields present results associated with the enforcement of the generator MW limit constraints.

#### Number of Initial Violations, MW Sum of Initial Violations

Total number of generators that initially exceeded their MW limits and were eligible for enforcement by the OPF. For these generators only, the **MW Sum of Initial Violations** field contains the sum of the absolute values of the generators' actual MW output minus the generators' MW limits.

**Number of Unenforceable Violations**

Total number of generators whose MW limits can not be enforced by the OPF using the available controls.

**MW Sum of Unenforceable Violations**

For all the generators with unenforceable limits, this field contains the sum of the absolute values of the generators' actual MW output minus the generators MW limits.

**Transformer Regulation Constraints**

The **Transformer Regulation Constraints** fields present results associated with the enforcement of phase shifting transformer regulation.

**Number of Initial Violations**

Total number of phase shifting transformers that initially exceeded their MW set point limits and were eligible for enforcement by the OPF.

**Number of Binding Constraints**

Total number of phase shifting transformers that are set at their minimum or maximum regulation value.

**Number of Unenforceable Violations**

Total number of phase shifting transformers whose MW regulation limits can not be enforced by the OPF using the available controls.

**Fast Start Generators**

The **Fast Start Generators** fields present results associated with the commitment/decommitment of fast start generators.

**Number of Generators Turned On**

Total number of fast start generators turned on during the OPF solution

**Number of Generators Turned Off**

Total number of fast start generators turned off during the OPF solution

---

**OPF Options: All LP Variables**


---

The OPF Dialog, All LP Basic Variables page displays the basic and non-basic variables associated with the final LP solution. This page is usually only of interest to users interested in the specifics of the LP solution. Right click anywhere in the display to copy a portion or all of the display to the Window's clipboard, or to print the results.

The display lists each of the LP variables, showing the following fields for each:

**ID**

Variable identifier.

**Original Value**

The initial value of the LP variable before OPF optimization.

**Value**

The final value of the LP variable after OPF optimization.

**Delta Value**

The difference between the original value field and the value field.

**BasicVar**

Shows the index of the basic variables in the LP basis. If the value is zero, the variable is non-basic.

**NonBasicVar**

Shows the index of the non-basic variable. If the value is zero, the variable is basic.

**Cost(Down)**

The cost associated with decreasing the LP variable. The field will also show if the variable is at its max or min limit.

**Cost(Up)**

The cost associated with increasing the LP variable. The field will also show if the variable is at its max or min limit.

**Down Range**

The available range to decrease the basic variable before a new constraint is hit.

**Up Range**

The available range to increase the basic variable before a new constraint is hit.

**Reduced Cost Up**

The cost reduction that would be experienced if an LP variable increases. If a constraint is at the limit, the field shows the change in cost of constraint enforcement.

**Reduced Cost Down**

The cost reduction that would be experienced if an LP variable decreases. If a constraint is at the limit, the field shows the change in cost of constraint enforcement.

**At Breakpoint?**

Yes, if the LP variable is at a break point.

---

**OPF Options: LP Basic Variables**

---

The OPF Dialog, LP Basic Variables page displays the basic variables associated with the final LP solution. This page is usually only of interest to users interested in the specifics of the LP solution. Right click anywhere in the display to copy a portion or all of the display to the Window's clipboard, or to print the results.

The display lists each of the basic variables, showing the following fields for each:

**ID**

Basic variable identifier.

**Original Value**

The initial value of the basic LP variable before the OPF optimization.

**Value**

The final value of the basic LP variable after the OPF optimization.

**Delta Value**

The difference between the original value field and the value of the basic variable.

**Basic Var**

Shows the indices of the basic variables in the LP basis.

**Cost(Up)**

The cost associated with increasing the basic variable.

**Down Range**

The available range to decrease the basic variable before a new constraint is hit.

**Up Range**

The available range to increase the basic variable before a new constraint is hit.

---

**OPF Options: LP Basis Matrix**

---

The OPF Dialog, LP Basis Matrix page displays the basis matrix associated with the final LP solution. This page is usually only of interest to users interested in knowing the specifics of the LP solution. Knowing the basis matrix can be helpful in figuring out why a particular power system is exhibiting a particular behavior. The rows of the basis matrix are the binding constraints, while the columns of the basis matrix are the basic variables. The entries in the basis matrix then give the sensitivity of each constraint to each of the basic variables. The width of the columns in the matrix can also be adjusted using the **Column Widths** field. Finally, you can right-click anywhere in the matrix to copy the matrix to the Window's clipboard or to print the matrix.

---

**OPF Options: Bus MW Marginal Price Details**

---

The Bus MW Marginal Price Details page displays a grid containing the MW marginal prices computed for an OPF solution. If no OPF solution has been run, the values will all be zero. The grid used for displaying the information is a Case Information Display, which can be modified, sorted, printed, etc., as described in the discussion of [Case Information Displays](#).

One key feature of this display is the breakdown in the bus MW marginal cost into three components: the cost of energy, the cost of congestion, and the cost of losses. Also, while the bus MW marginal cost does not change based on the reference, the costs of energy, congestion, and losses are reference dependent. This reference is set on a per-area (or per-super area) basis in the [Area information dialog OPF tab](#) (or the [Super Area information dialog OPF tab](#)).



---

**OPF Options: Bus MVAR Marginal Price Details**

---

The Bus MVAR Marginal Price Details page displays a grid containing the MVAR marginal prices computed for an OPF solution. If no OPF solution has been run, or if the option to compute MVAR marginal prices has not been selected, the values will all be zero. The grid used for displaying the information is a Case Information Display, which can be modified, sorted, printed, etc., as described in the discussion of [Case Information Displays](#).

---

**OPF Options: Bus Marginal Controls**

---

This display shows the sensitivities of the controls with respect to the cost at each bus. A change in a system control will have the indicated effect in the marginal cost at the system buses. Vice-versa, the marginal cost at a bus is affected by changes in the value of the basic variables.

---

**OPF Options: Inverse of LP Basis**

---

The OPF Dialog, LP Basis Matrix page displays the inverse of the basis matrix. The width of the columns in the matrix can also be adjusted using the **Column Widths** field. Finally, right-click somewhere in the matrix to copy the matrix to the Window's clipboard or to print the matrix.

---

**OPF Area Records**

---

Displays OPF specific information about each area record in the case. To show this display select **Optimal Power Flow > Results > Areas** in the [Model Explorer](#). The OPF Area Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the local menu, which can be accessed by right-clicking on any field in the display.

By default the display contains the following fields:

**Number, Name**

Area's number, between 1 and 999, and its alphanumeric identifier, eight characters maximum.

**AGC Status**

Area's automatic generation control status. This is the same field shown on the [Area Records Display](#). The field indicates whether or not the area's generation is changing automatically to control the area interchange.

To be included in the OPF, this field MUST be "OPF". The generation costs for areas that are on "OPF" control are included in the [OPF objective function](#); otherwise they are not. Note that if the area is part of a super area that is on AGC control, this field value is ignored.

Double-click on the field to toggle its value.

**XF Phase**

Specifies whether phase shifting transformers in the area are available as controls. If "Yes" then all transformers in the area which have their Automatic Control Active are available for control; the Automatic Control status for a transformer is set on the [Line/Transformer Dialog](#). If "No" then no transformers in the area are available for control.

**Branch MVA**

Specifies whether or not the MVA limits should be enforced for transmission lines and transformers that have at least one terminal in this area. For a transmission line or transformer to be included Line/Transformer constraints must not be disabled on the OPF Options and Results Dialog, and the individual line/transformer must be enabled for enforcement on the OPF Line/Transformer MVA Constraints display.

**Interface MW**

Specifies whether or not the MW limits should be enforced for interfaces that have at least one element in this area. For an interface to be included Interface constraints must not be disabled on the OPF Options and Results Dialog, and the individual interfaces must be enabled for enforcement on the OPF Interface MW Constraints display.

**Load MW Dispatch**

Specifies whether or not the MW load demand in an area should be included as available for re-dispatch during an OPF solution. In order for loads to be included in OPF re-dispatch, each individual load within the area must be available for control, and have either a fixed cost benefit or a piecewise-linear cost benefit curve provided.

**DC Line MW Control**

Specifies whether or not DC line MW setpoints can be modified for DC lines that have at least one terminal in this area.

**Include Marg. Losses**

Specifies whether or not marginal losses should be included for the area during the OPF solution.

**MW Marg. Cost Ave**

For an OPF solved case this field shows the average of the bus MW marginal costs for all the buses in the area. If there is no congestion then all of the marginal costs should be equal.

**MW Marg. Cost St.Dev., Min., Max.**

For an OPF solved case these fields show the standard deviation of the bus MW marginal costs for all the buses in the area, the minimum and the maximum bus MW marginal costs.

**Report Limits**

Specifies whether or not the kV limits should be reported.

**Report Min kV, Report Max kV**

Specifies the values for minimum and maximum kV levels to report. Defaults are 0 and 9999.

---

**OPF Bus Records**

---

Displays OPF specific information about each bus record with a valid [area/zone/owner filter](#). To show this display select **Optimal Power Flow > Results > Buses** in the [Model Explorer](#). The OPF Bus Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the [local menu](#), which can be accessed by right-clicking on any field in the display. The columns can also be sorted by right-clicking on the heading of the field.

By default the display contains the following fields

**Number, Name**

Bus's number, between 1 and 2,147,483,647 (equals  $2^{31}$  minus 1), and its alphanumeric identifier.

**Area Name**

Name of the bus's area.

**MW Marg. Cost**

Marginal change in the objective function for a one MW change in the real power load at the bus.

**MVR Marg. Cost**

Marginal change in the objective function for a one Mvar change in the reactive load at the bus.

**Volt Marg. Cost**

Marginal change in the objective function for a 0.01 per unit change in the voltage setpoint for the bus. This field is only valid at bus's whose terminal voltage is controlled by one or more generators.

*Note from the developers – this field is still under construction – do not use it yet.*

---

## OPF Generator Records

---

Displays OPF specific information about each generator record with a valid [area/zone/owner filter](#). To show this display select **Optimal Power Flow > Results > Generator Records** in the [Model Explorer](#). The OPF Generator Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the [local menu](#), which can be accessed by right-clicking on any field in the display.

By default the display contains the following fields:

### Number, Name

Number and name of the bus to which the generator is attached. The display's local menu offers you the opportunity to view the [Quick Power Flow List](#) and the [Bus View Display](#) for this bus. You can also use the local menu to view the [generator's dialog](#).

### ID

Alphanumeric ID used to distinguish multiple generators at the same bus.

### Area Name of Gen

Name of the generator's area.

### AGC

Designates whether the generator's real power output is governed by automatic generation control (AGC). If the AGC field is set to Yes the generator is on AGC in the standard power flow. When a generator is on AGC its real power output is varied automatically, provided the generator is part of an area or super area that is also on automatic control.

In Simulator OPF the default operating mode is that only generators on AGC control are eligible to be OPF controls. In addition, the generator's area or super area must have **AGC Status** of "OPF". However in rare instances you may wish to always make a generator available for control or never make the generator available for control. This value is specified using **OPF MW Control** field.

### Fast Start

Designates whether the generator is available as a Fast Start generator during the OPF solution process. Fast start generators are another type of control available to the Optimal Power Flow solution routine. The OPF routine can determine if a generator labeled as a fast start generator would be beneficial in reducing the overall system costs of generation dispatch. If a fast start generator is off-line, but could reduce the cost of the system, then the OPF routine will turn on the generator, and increase the generator's dispatch towards optimizing the system generating cost. Conversely, if a fast start generator is on-line, and the OPF routine determines that reducing the generator's output to 0 would reduce the total generation cost, then the OPF routine will shut off the generator.

Generally speaking, the fast start options should only be used with units with zero Minimum MW limits. Hence it is really aimed at hydro units, or small units which do not need a non-zero minimum MW output for valid operation. This requirement is needed because changing a unit's status is only valid in the OPF routine if it is determined that the unit should dispatch 0 MW to optimize the generating costs of the system.

### OPF MW Control

Designates whether the generator's real power output should be included as a control variable in the OPF. This field, which can be toggled, has three possible values:

- "If AGCable" - Generator's control availability depends upon its AGC status.

- "Yes" - Generator is available as a control, regardless of its AGC status.
- "No" - Generator is NOT available as a control, regardless of its AGC status.

**Note:** in order to be a control the generator must also be in an area or super area on "OPF" control.

#### **Gen MW**

The real power output of the generator.

#### **Cost Shift \$/MWh, Cost Multiplier**

The cost shift and cost multiplier allow you to easily apply a shift to the cost function for the purpose of assessing how variations in bids impact profit. The cost function is affected based on the following equation:

$(\text{Original Cost Function} + \text{Cost Shift}) * \text{Cost Multiplier}$

#### **Cost \$/Hr**

The total cost of the generator, including the impact of the cost shift and cost multiplier.

#### **MW Marg. Cost**

Tells the marginal cost, in \$ / MWhr, to supply one additional MW of load at this bus. If a generator is available as a control and is not at either its minimum or maximum limit or a cost model breakpoint, then the **MW Marg. Cost** field will be identical to the generator's current marginal cost. However the usual case is for the generator to be at either a limit or a cost model breakpoint so the usual situation is that the **MW Marg. Cost** field values IS NOT equal to the generator's marginal cost.

#### **IC for OPF**

Incremental cost of the generator at its current operating point.

#### **Initial MW**

The initial real power output of the generator at the beginning of the OPF solution. You can reset the case back to these values by selecting the LP OPF, Restore Previous Control Settings menu item. This menu item is only available following a successful OPF solution.

#### **Initial Cost**

The initial generator cost at the beginning of the OPF solution.

#### **Delta MW**

Change in the generator's real power output as a result of the OPF.

#### **Delta Cost**

Change in the generator's cost as a result of the OPF.

#### **Min MW, Max MW**

Minimum and maximum real power output of the generator.

#### **Cost Model**

The current cost model being used for the generator. The field value is either "Cubic", indicating that the generator's operating costs are being modeled using a cubic cost function, or "Piecewise Linear", indicating the operating costs are being modeled using a piecewise linear cost function. Toggle the field to change the model. Note that a generator may simultaneously have a cubic model and a piecewise linear model.

Because the OPF uses a linear programming approach, the generator's operating costs are ALWAYS modeled using the piecewise linear model. Generators with an existing cubic cost model are either 1) ignored as OPF controls, or 2) have a piecewise linear cost model



automatically created from the cubic model, depending upon the values specified on the [OPF Options and Results Dialog](#).

**# Cost Curve Points**

Shows the number of segments in the piecewise linear model. If no piecewise linear model exists then this field is zero; the generator's costs are being modeled using the cubic function. For such generators you can automatically setup a piecewise linear model simply by entering a non-zero value for the number of points. A piecewise linear model is created that matches as closely as possible the existing cubic model.

**Fuel Type**

Specifies the fuel type of the generator, if it is known; double-click to toggle through the options. Options are Unknown, Coal, Gas, Hydro, Hydro Pumped, Nuclear, Petroleum, Solar, Wind, and Other.

**Profit \$/hr**

Shows the profit of the generator. Profit is calculated using this equation:

$$\text{Profit} = (\text{GenMW} * \text{MW Marg Cost}) - [\text{Evaluation of the Generator Cost Function}]$$

---

## OPF Interface Records

---

Displays OPF specific information about the interface records in the case. To show select **Optimal Power Flow > Results > Interfaces** from the [Model Explorer](#). This display is actually a page of a display showing all the potential inequality constraints in the power system. The top portion of the display repeats the fields shown on the [Constraint Options](#) page of the [OPF Options and Results dialog](#).

To view the Interface records click on the Interfaces tab. This displays the interfaces page which is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the [local menu](#), which can be accessed by right-clicking on any field in the page.

By default the Interface Records page contains the following fields:

### **Name**

Name of the interface.

### **Monitor**

Specifies whether or not the interface MW limit is enforced in the OPF solution.

### **Interface MW**

The amount of MW flow on the interface.

### **MW Limit**

The interface MW limit.

### **Percent**

The amount of MW flow on the interface as a percentage of the limit.

### **Monitor Direction**

The direction on the interface in which the flow is being monitored.

### **Monitor Both Directions**

If set to **YES**, then the interface flow will be monitored in both directions. Otherwise, it will only be monitored in the **Monitor Direction**.

### **MW Marg. Cost \$ / MWh**

The marginal cost of enforcing the limit on the interface.

### **Constraint Status**

This field will be set to Binding if the interface limit is a binding constraint in the OPF solution, or Unenforceable if the interface limit is unenforceable with the available control set.

---

## OPF Nomogram Records

---

This list displays OPF specific information about the nomogram records in the case. To show the list, select **Optimal Power Flow > Results > Nomograms** in the [Model Explorer](#). This display is actually a page of a display showing all the potential inequality constraints in the power system. The top portion of the display repeats the fields shown on the [Constraint Options page](#) of the [OPF Options and Results dialog](#).

To view the nomogram records click on the Nomogram Interfaces tab. This displays the nomogram interfaces page which is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the [local menu](#), which can be accessed by right-clicking on any field in the page.

By default the Interface Records page contains the following fields:

### **Nomo. Name**

Name of the nomogram.

### **Nomo. Seg.**

The segment of the nomogram. Each segment of the nomogram is treated as a separate constraint in the OPF, and therefore is listed individually in the OPF nomogram table.

### **Monitor**

This field will be set to YES if the nomogram segment is monitored during the OPF.

### **Interface MW Flow**

The total MW flow for the segment of the nomogram.

### **MW Limit**

The segment MW limit.

### **Percent**

The amount of MW flow on the segment as a percentage of the segment's limit.

### **Monitor Direction**

The direction on the segment in which the flow is being monitored.

### **Monitor Both Directions**

Yes or No field indicating if both directions should be monitored at the same time.

### **MW Marg. Cost \$ / MWh**

The marginal cost of enforcing the limit on the segment.

### **Constraint**

This field will be set to Binding if the nomogram limit is a binding constraint in the OPF solution, or Unenforceable if the nomogram limit is unenforceable with the available control set.

---

**OPF Line/Transformer Records**


---

Displays OPF specific information about the line and transformer records in the case. To show the display select **Optimal Power Flow > Results > Branches** in the [Model Explorer](#). This display is actually a page of a display showing all the potential inequality constraints in the power system. The top portion of the display repeats the fields shown on the [Constraint Options page](#) of the [OPF Options and Results dialog](#).

To view the Line and Transformer records click on the Line/Transformers tab. This displays the Line/Transformer page which is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the [local menu](#), which can be accessed by right-clicking on any field in the display.

By default the display contains the following fields:

**From Number, From Name, From Area Name**

Number, Name, and Area Name of the From bus.

**To Number, To Name, To Area Name**

Number, Name, and Area Name of the To bus.

**Circuit**

Circuit identifier for the branch.

**Monitor**

Specifies whether or not the branch's MVA limit will be enforced in the OPF solution.

**Max MVA**

The maximum MVA flow on the branch. Value is determined based on the end of the branch with the higher MVA flow.

**% of MVA Limit (Max)**

The maximum MVA flow as a percentage of the branch MVA limit. Value is determined based on the end of the branch with the higher MVA flow.

**Lim MVA**

The MVA limit for the branch.

**MVA Marg. Cost (\$/MVAhr)**

The marginal cost of changing the MVA flow on the branch (i.e., if the line rating were to increase by 1 MVA, what savings could be had).

**Constraint**

This field will be set to Binding if the branch limit is a binding constraint in the OPF solution, or Unenforceable if the branch limit is unenforceable with the available control set.

---

**OPF DC Lines Records**

---

Displays OPF specific information about the DC lines in the case. To show select **Optimal Power Flow > Results > DC Lines** in the [Model Explorer](#). This display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the [local menu](#), which can be accessed by right-clicking on any field in the display.

By default the display contains the same fields as the normal [DC Line Records](#) case information display, with the following additional fields:

**OPF Control**

Set to **YES** if the DC line is to be controlled during the OPF. To be controlled, at least one of the terminals of the DC line must lie in an area that is on OPF control and this must be set to **YES**.

**Min MW or amps, Max MW or amps**

The minimum and maximum power (or current) setpoints allowed for the DC line.

---

## OPF Load Records

---

Displays OPF specific information about each load record with a valid [area/zone/owner filter](#). To show this display select **Optimal Power Flow > Results > Loads** in the [Model Explorer](#). The OPF Load Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the [local menu](#), which can be accessed by right-clicking on any field in the display.

By default the display contains the following fields:

### Number, Name

Number and name of the bus to which the load is attached. The display's local menu offers you the opportunity to view the [Quick Power Flow List](#) and the [Bus View Display](#) for this bus. You can also use the local menu to view the [generator's dialog](#).

### ID

Alphanumeric ID used to distinguish multiple loads at the same bus.

### Area Name of Load

Name of the load's area.

### AGC

Designates whether the load's real power demand is governed by automatic generation control (AGC). If the AGC field is set to Yes the load is on AGC in the standard power flow. When a load is on AGC its real power output is varied automatically during an OPF or SCOPF solution ONLY, provided the load is part of an area or super area that is also on OPF control. Loads will not be dispatched if the area is on some other form of AGC control.

In Simulator OPF the default operating mode is that only loads on AGC control are eligible to be OPF controls. In addition, the load's area or super area must have **AGC Status** of "OPF", and the area must have its Load MW Dispatch set to YES.

### MW

The real power output of the load.

### Cost Shift \$/MWh, Cost Multiplier

The cost shift and cost multiplier allow you to easily apply a shift to the cost function for the purpose of assessing how variations in bids impact profit. The cost function is affected based on the following equation:

$$(\text{Original Cost Function} + \text{Cost Shift}) * \text{Cost Multiplier}$$

### Hourly Benefit

The total benefit of the load, including the impact of the cost shift and cost multiplier.

### MW Marg. Cost

Tells the marginal cost, in \$ / MWhr, to reduce one additional MW of load at this bus. If a load is available as a control and is not at either its minimum or maximum limit or a cost model breakpoint, then the **MW Marg. Cost** field will be identical to the load's current marginal cost. However the usual case is for the load to be at either a limit or a cost model breakpoint so the usual situation is that the **MW Marg. Cost** field value IS NOT equal to the load's marginal cost.

### Inc. Benefit

Incremental benefit of the load at it's current operating point.

**Initial MW**

The initial real power demand of the load at the beginning of the OPF solution.

**Initial Cost**

The initial load cost at the beginning of the OPF solution.

**Delta MW**

Change in the load's real power output as a result of the OPF.

**Delta Cost**

Change in the load's cost as a result of the OPF.

**Min MW, Max MW**

Minimum and maximum real power demand of the load.

**# of Benefit Curve Points**

Shows the number of segments in the piecewise linear model. If no piecewise linear model exists then this field is zero.

**Profit \$/hr**

Shows the profit of the load.

---

**OPF Super Area Records**


---

Displays OPF specific information about each super area record in the case. To show this display select **Optimal Power Flow > Results > Super Areas** in the [Model Explorer](#). The OPF Super Area Records Display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the local menu, which can be accessed by right-clicking on any field in the display.

By default the display contains the following fields:

**Super Area**

Alpha-numeric identifier of the super area.

**AGC Status**

Super area's automatic generation control status. This is the same field shown on the Super Area Records display. The field indicates whether or not the super area's generation is changing automatically to control the super area's interchange.

The super area AGC Status field always overrides the AGC Status for the individual areas, except when it is set to "Off AGC".

To be included in the OPF this field MUST be "OPF". In that case the generation costs for all the areas in the super area are included in the OPF objective function. Otherwise they are only included if the super area AGC Status is "Off AGC" and their particular area's AGC Status is on "OPF". Double-click on the field to toggle its value.

**Num Areas**

Number of areas in the super area. To see the individual areas use the local menu to view the Super Area dialog.

**Include Marg. Losses**

Specifies whether or not to include marginal losses in OPF calculations for this super area.

**MW Marg. Cost Ave.**

If the super area is on "OPF" control then for a solved case this field shows the average of the bus MW marginal costs for all the buses in the super area. If there is no congestion then all of the marginal costs should be equal.

**MW Marg. Cost St. Dev.**

Standard deviation of the bus MW marginal costs for the buses in the super area.

**ACE MW**

Area control error for the super area.

**Gen MW, Load MW**

Total real power generation and load in the super area.

**Total Sched MW, Int MW**

Scheduled and actual interchange real power interchange between the super area and the rest of the system. Both of these fields are the algebraic summation of the scheduled and actual interchange for the areas in the super area.

**Loss MW**

Total real power losses for the super area.



**MW Marg. Cost Min, MW Marg. Cost Max**

Minimum and maximum of all the bus MW marginal costs for the buses in the super area.

---

## OPF Controls

---

The following classes of controls are available during the OPF solution. Note, individual classes of controls can be enabled/disabled for the entire case using the OPF Options and Results dialog and for particular areas using the OPF Area Records display. Also, all classes of controls have associated minimum/maximum limits which are always enforced.

### Generator MW output and Fast Start commitment

The generator MW outputs are the major control for controlling the MW flow in the network and for minimizing the objective function. Only generators in areas or super areas that are on "OPF" control are eligible for control; otherwise the generator's MW output remains fixed at its initial value. Whether a particular generator is available for control also depends upon the status of its AGC and OPF MW Control fields. These fields are set on the [OPF Generator Records](#) display. For Fast Start Generators, it is also possible to use the startup and shutdown of these generators as a control. See [OPF Options: Control Options](#) for more details.

### Phase shifting Transformer tap position

Phase shifting transformers are used primarily to control the flow of real power in the network. When phase shifting transformers are controlled in the OPF routine, the phase angle is allowed to move anywhere within the phase angle range of the device in order to help alleviate violations on other branches in the system. The flow on the phase shifter is allowed to violate the prescribed MW range given for the phase shifter, but is NOT allowed to violate the MVA rating of the device.

Additionally, in order for a phase shifter to be included in the OPF solution dispatch, the XF Phase property of the area must be set to YES. Each individual phase shifter also has a property for being included in the OPF control that must be turned on in order for the phase shifter to participate in the OPF dispatch. This option can be set for a phase shifting transformer by opening its [Information Dialog](#) and checking the OPF Phase Shifter Control options on the OPF page of the dialog.

### Load MW Dispatch

The load MW demands can also be included as controls for re-dispatch during an OPF solution. The concept of controlling a load is generally the same as controlling a generator. Loads can be assigned piecewise linear benefit curves, and included in the OPF dispatch algorithm. Only areas whose AGC control is set to "OPF" are eligible for control.

Furthermore, the OPF area's Load MW Dispatch property must also be set to YES. Each area load that is to be included for OPF dispatch must have a benefit model defined, and have its Available for AGC field set to YES. These fields can be set in the [OPF Load Records](#) display.

### DC Line MW setpoint

DC Line MW setpoints can be used as controls in the OPF. See [OPF DC Lines Records](#).

---

**OPF Phase Shifter Records**


---

Displays OPF specific information about each phase shifter record in the case. To show this display, select **Optimal Power Flow > Results > Phase Shifters** in the [Model Explorer](#). The OPF Phase Shifter Records display is a class of [Case Information Display](#) and therefore can be used in a manner consistent with the other case information displays. Specific formatting options are available from the local menu, which can be accessed by right-clicking on any field in the display. The columns can also be sorted by right-clicking on the heading of the field.

By default the display contains the following fields:

**From Number, From Name**

The name and number of the bus at the From end of the phase shifter.

**To Number, To Name**

The name and number of the bus at the To end of the phase shifter.

**Circuit**

The circuit identifier for the phase shifter.

**OPF Control**

Specifies whether or not the phase shifter is available for control during an OPF solution.

**Area PS Control**

Specifies if automatic phase shifter control has been enabled for the area containing the particular phase shifting transformer. If the area phase shifter control is disabled, all phase shifting transformers within the area will remain fixed at their initial settings during the entirety of the OPF solution process. This setting overrides the individual automatic control settings of each phase shifting transformer within the area.

**XF Auto**

Specifies if the transformer automatic control is enabled. If an individual transformer's automatic control is disabled, it will remain at its initial settings during the entirety of the OPF solution process.

**Phase (Deg)**

The actual phase shift of the phase shifter, in degrees.

**Initial Degrees**

The initial phase angle before the OPF solution was calculated.

**Delta Degrees**

The change in the phase angle during the OPF solution.

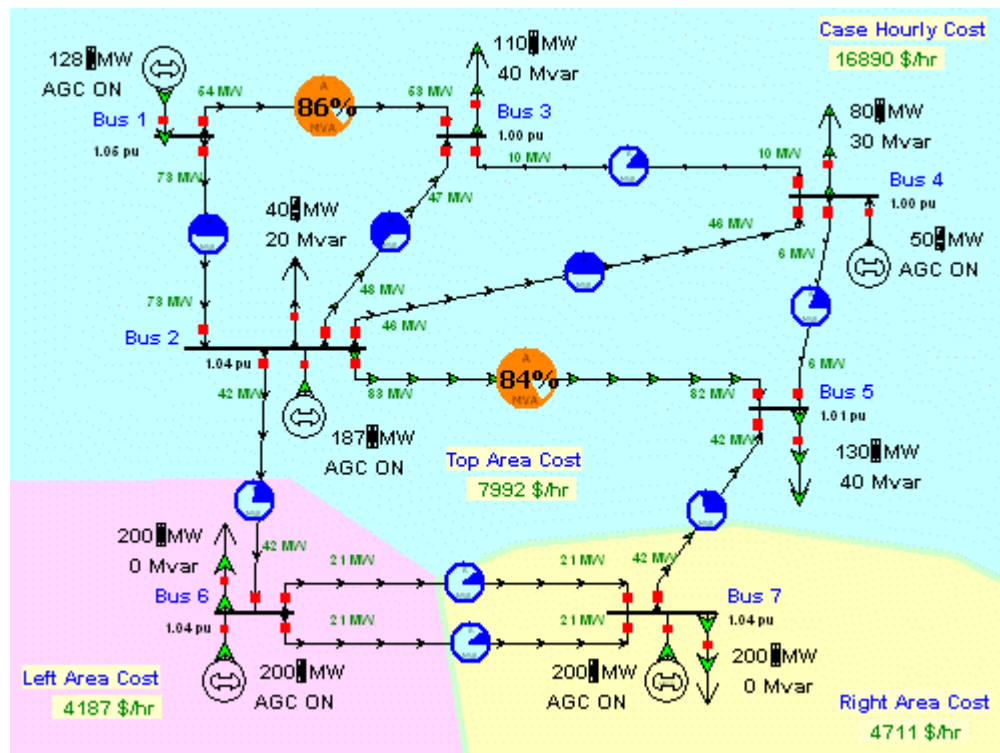
**Tap Min, Tap Max**

The minimum and maximum tap positions allowed for the phase shifter operation.

## OPF Example - Introduction

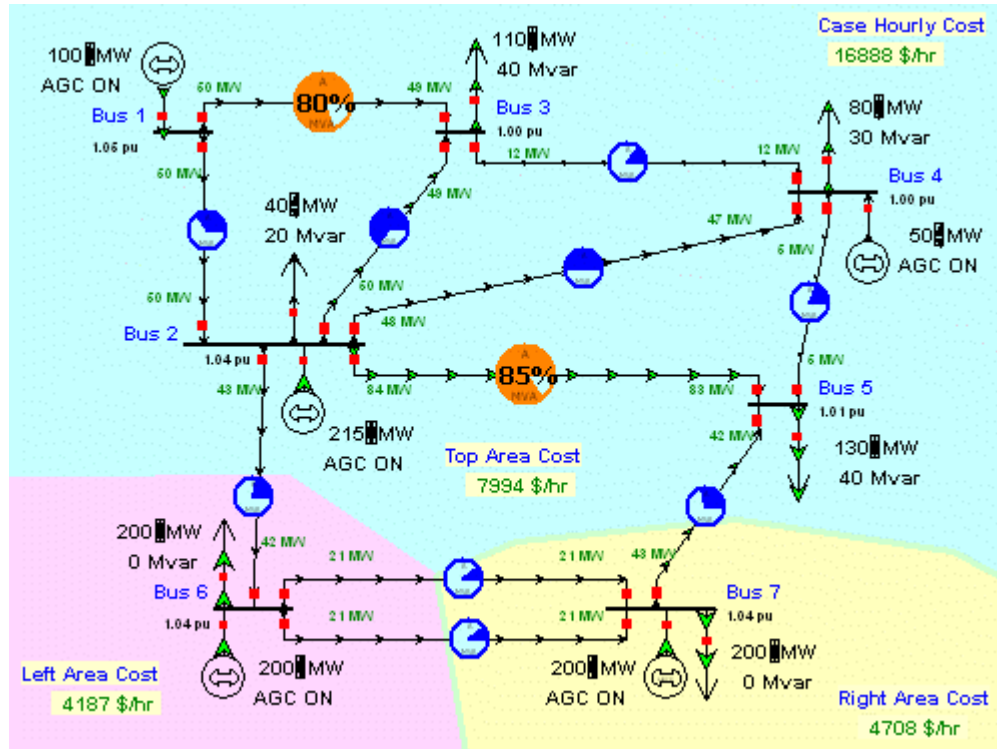
As a simple example of using the OPF, consider the seven bus, three area system contained in the file B7OPF.pwb (included with the PowerWorld Simulator). For this case all three areas are initially on Economic Dispatch (ED) AGC control and hence by default would not be included in the OPF solution. Also, the initial interchange between the areas is equal to zero and the generators are modeled using cubic cost functions.

To initially solve the case using the standard power flow, select **Single Solution** in the [Power Flow Tools](#) ribbon group on the [Tools](#) ribbon tab. The case should look like the following figure:



B7OPF Case Solved using Economic Dispatch

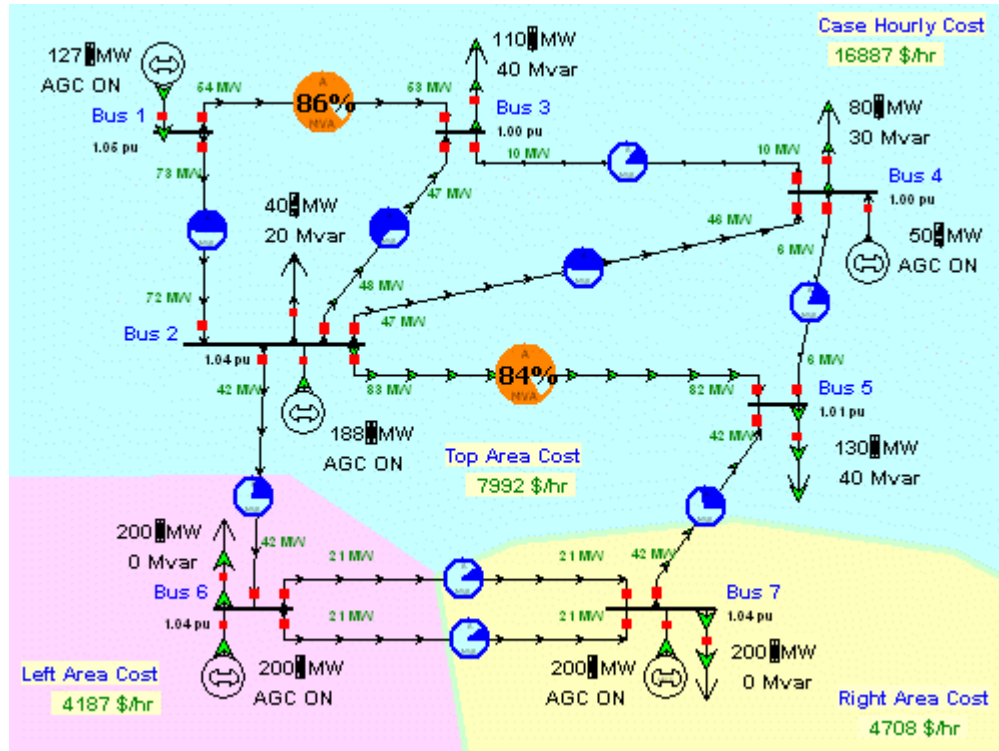
Now we'll modify the case to set the three areas for OPF control. To do this, select **Optimal Power Flow > Results > Areas** in the [Model Explorer](#) to display the [OPF Area Records display](#). Toggle the AGC status for each of the three areas to change it to "OPF". Now select **Primal LP** in the **OPF** ribbon group on the [Add Ons](#) ribbon tab to solve the case using the LP OPF. The results should look similar to the figure below. The one-line shows the hourly cost for each area and the total case hourly cost, equal to \$ 16,888 / hr.



B7OPF Case Solved using LP OPF

Note that the results are very similar but not identical to the economic dispatch case. We would expect the cases to be similar since for cases with no congestion the OPF solution should be (ideally) equal to the economic dispatch solution. The difference between the two is because in the LP OPF the generator cost functions are converted from a cubic model to a piece-wise linear model using a user specified number of segments, which is 5 segments by default. This value can be viewed/modified from the Control Options page of the [OPF Options and Results display](#).

Change the **Total Points Per Cost Curve** field to 100 and resolve. The results are shown in the figure below, which now are almost identical to the economic dispatch results. The disadvantage in using a large number of cost segments is that it degrades the performance of the LP OPF slightly on larger cases.



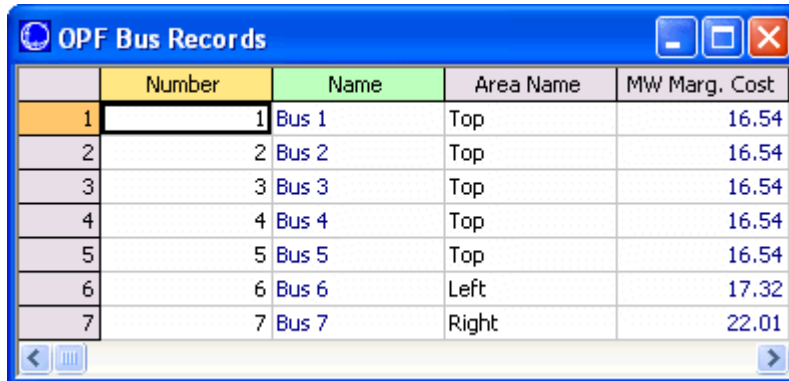
B7OPF Case Solved using LP OPF with 100 Cost Segments for Generators

---

**OPF Example - Marginal Costs**

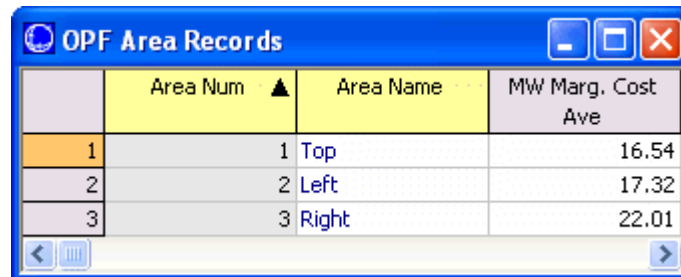

---

Using the OPF solution from the previous page, select **Optimal Power Flow > Results > Buses** in the [Model Explorer](#) to view the bus marginal costs and **Optimal Power Flow > Results > Areas** in the [Model Explorer](#) to view the area marginal costs. The results should be as shown below.



	Number	Name	Area Name	MW Marg. Cost
1	1	Bus 1	Top	16.54
2	2	Bus 2	Top	16.54
3	3	Bus 3	Top	16.54
4	4	Bus 4	Top	16.54
5	5	Bus 5	Top	16.54
6	6	Bus 6	Left	17.32
7	7	Bus 7	Right	22.01

Seven Bus Case Bus Marginal Costs



	Area Num ▲	Area Name	MW Marg. Cost Ave
1	1	Top	16.54
2	2	Left	17.32
3	3	Right	22.01

Seven Bus Case Area Marginal Costs

Note that the marginal costs for all the buses in an area are identical to the area's MW marginal cost. This is the expected result for systems without any line congestion. The area MW marginal costs are not identical. This is because currently each area is independently enforcing its own MW interchange. In the next example we'll jointly dispatch the three areas by combining them into a single [super area](#).

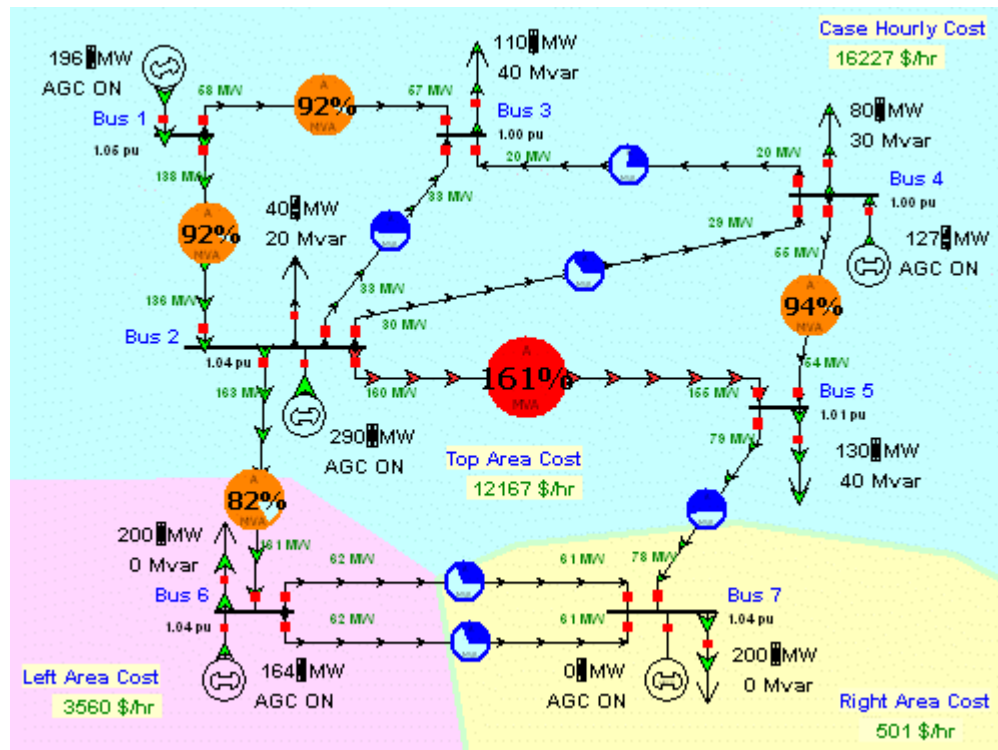
## OPF Example - Super Areas

To jointly dispatch the three areas we'll first combine them into a single super area. To setup the super area first select **Optimal Power Flow > Results > Super Areas** from the [Model Explorer](#) to display the [OPF Super Area Records display](#) (alternatively you could also use the **Aggregations > Super Areas** display). To enter a new super area, right click on the "None Defined" entry in the first row of the display to show the display's local menu. Select **Insert**. This displays the [super area dialog](#). Select **Rename** and enter a name for the new super area, "ThreeAreas." To add the three areas to the new super area enter "1-3" in the **New Area #'s** field and then select **Add New Areas by Number** (alternatively you could select the areas from the **New Area Name** list). Also, to enable the super area for control, set the **Super Area Control Options** field (AGC Status) to Optimal Power Flow Control. Select the **Ok** button to save the new super area.

Before resolving the OPF, let's temporarily disable enforcement of line MVA constraints. You can do this from the [Constraint Options page](#) of the [OPF Options and Results dialog](#). Check the **Disable Line/Transformer MVA Limit Enforcement**.

Also, now would be a good time to save the changes. To avoid overwriting the existing B7OPF file, select **Save Case As** from the [Application button](#) to save the case (pwb and pwr files) with a different name, say B7OPFSA (SA for super area).

Select **Primal LP** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab to resolve the OPF. The results should be as shown below.



OPF Solution with Super Area WITHOUT Enforcing Line MVA Constraints

With the super area the individual area interchange constraints are no longer enforced. This permits the free interchange of power between the areas, resulting in an overall decrease in the total case hourly cost from \$ 16,887 / hr to \$ 16,227 / hr. View the OPF Bus Records display to verify that all the bus marginal costs are identical, equal to \$ 17.10 / MWh. Of

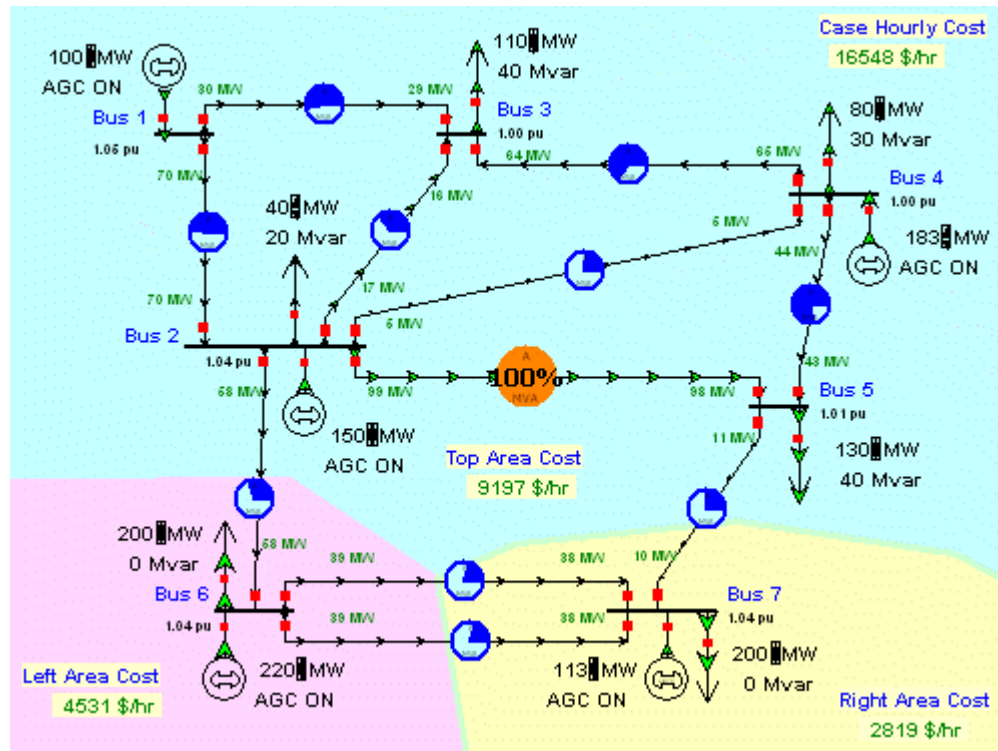


course the key problem with solving the system using the super area is that now there are line violations. These violations will be removed next by enforcing the line MVA constraints.

## OPF Example - Enforcing Line MVA Constraints

To remove the line MVA violations go back to the [Constraint Options Page](#) of the [OPF Options and Results](#) dialog. Uncheck the **Disable Line/Transformer MVA Limit Enforcement**.

Now resolve the LP OPF, enforcing the line constraints. The resultant solution is shown below.



OPF Solution with Super Area WITH Line MVA Constraint Enforcement

With line constraint enforcement active the OPF optimally redispatches the generation taking into account the line MVA limits. However, enforcing these line constraints comes at a cost. Notice that the total case hourly cost has increased from \$ 16,227 / hr to \$ 16,548 / hr, which is still substantially less than the \$ 16,887 / hr figure we had for the case without the superarea.

Enforcing the line constraints also has an impact on the bus marginal costs, shown below.

	Number	Name	Area Name	MW Marg. Cost
1	1	Bus 1	Top	15.97
2	2	Bus 2	Top	15.63
3	3	Bus 3	Top	17.00
4	4	Bus 4	Top	17.42
5	5	Bus 5	Top	23.07
6	6	Bus 6	Left	17.44
7	7	Bus 7	Right	21.11

Impact of Line MVA Enforcement on the Bus Marginal Costs

The actual marginal cost of enforcing the line constraint can also be viewed on the [OPF Line/Transformer MVA Records](#) display. The MVA Marg. Cost tells the marginal cost of enforcing the constraint, expressed in units of \$ / MVA / hr. Left-click on the **MVA Marg Cost** field header to sort the display using this field.

**OPF Constraints Records**

**Line/Transformer Constraints**

☐ Disable Line/Transformer MVA Limit Enforcement

Percent Correction Tolerance: 2.0

☐ Only Show Limit Violations

☒ Only Areas with Line MVA Enforcement

☐ Use Area/Zone Filters

☐ Enforce Line/Transformer MW Flow Limits (not MVA)

**Interface Constraints**

☐ Disable Interface MW Limit Enforcement

Percent Correction Tolerance: 2.0

☐ Only Show Limit Violations

☐ Use Area/Zone Filters

**Monitor/Enforce Contingent Interface Elements**

☒ Never

☐ Power Flow/OPF but not CA/SCOPF

☐ All Applications including CA/SCOPF

☐ Do not enforce limits on radials lines and buses

Limit Monitoring Settings ...

If you want to change enforcement percentages, modify the Limit Monitoring Settings

Lines/Transformers | Interfaces

	From Number	From Name	From Area Name	To Number	To Name	To Area Name	Circuit	Monitor	Max MVA	% of MVA Limit (Max)	Lim MVA	MVA Marg Cost ▼
1	2	Bus 2	Top	5	Bus 5	Top	1	YES	100.0	100.0	100.0	13.8
2	6	Bus 6	Left	7	Bus 7	Right	1	YES	39.7	19.9	200.0	
3	1	Bus 1	Top	3	Bus 3	Top	1	YES	34.6	53.3	65.0	
4	3	Bus 3	Top	4	Bus 4	Top	1	YES	64.6	64.6	100.0	
5	2	Bus 2	Top	4	Bus 4	Top	1	YES	23.9	23.9	100.0	
6	7	Bus 7	Right	5	Bus 5	Top	1	YES	50.7	25.4	200.0	
7	2	Bus 2	Top	6	Bus 6	Left	1	YES	59.5	29.7	200.0	
8	6	Bus 6	Left	7	Bus 7	Right	2	YES	39.7	19.9	200.0	
9	1	Bus 1	Top	2	Bus 2	Top	1	YES	70.5	47.0	150.0	
10	4	Bus 4	Top	5	Bus 5	Top	1	YES	45.6	76.0	60.0	
11	2	Bus 2	Top	3	Bus 3	Top	1	YES	28.1	35.1	80.0	

---

## Security Constrained Optimal Power Flow Overview

---

**Note: The SCOPF option in PowerWorld Simulator is only available if you have purchased the SCOPF and OPF add-ons to the base package. To learn more about the SCOPF, please feel free to read through the information contained in these help files. Contact PowerWorld Corporation for details about ordering the SCOPF and OPF versions of Simulator.**

The [optimal power flow](#) (OPF) algorithm has the purpose of minimizing an objective function (usually total operation cost) by changing different system controls while meeting power balance constraints and enforcing base case operating limits. Normally however, the secure operation of a power system requires that there be no unmanageable contingency violations. Thus, the minimization of the objective function requires considering contingencies. This is achieved using a security constrained optimal power flow (SCOPF) algorithm.

While the [optimal power flow](#) (OPF) algorithm determines the optimal state of the system by iterating between solving a power flow and solving a linear program that changes the system controls, during the [SCOPF solution process](#) in addition to solving the base case power flow, all contingencies must also be solved. The linear program which then runs changes the system controls to remove contingency violations as well as base case operating violations.

The SCOPF algorithm makes control adjustments to the base case (pre-contingency condition) to prevent violations in the post-contingency conditions. If enough controls are available in the system, the solution minimizes the [objective function](#) and the system enforces contingency violations. If the system does not have enough controls, then some violations may be persistent under certain contingencies. Those represent unenforceable constraints, which result in high bus marginal costs.

The commands and options for the SCOPF are accessed by selecting the **SCOPF** option from the **OPF** ribbon group on the [Add Ons](#) ribbon tab. The SCOPF function uses the OPF options defined in the [OPF Options and Results dialog](#) and the contingency settings specified in the [contingency analysis dialog](#). Please read through the OPF and the contingency analysis help for more information about these settings. We encourage you to become familiar with Simulator OPF and the contingency analysis tool before running SCOPF simulations. Other commands in the [SCOPF dialog](#) are used to specify the base case solution process and for accessing the SCOPF results.

---

## SCOPF Objective Function

---

The SCOPF objective function uses the function defined in the OPF settings. There are two objective functions in Simulator: Minimum Cost and Minimum Control Change. Minimum Cost attempts to minimize the sum of the total generation costs in specified areas or super areas. Minimum Control Change attempts to minimize the sum of the absolute value of the change in the generation in the specified areas or super areas. The objective function is set up in the [OPF Options and Results dialog](#).

The result of the SCOPF will be different from the OPF solution because the SCOPF meets additional [inequality constraints](#) associated with the contingency violations.

---

## SCOPF Dialog

---

The SCOPF dialog allows the user to control options of Security Constrained OPF, as well as to access the optimization results.

The SCOPF can be run from this dialog using the **Run Full Security Constrained OPF** button at the top of the form.

There are three pages of information included in this form:

[Options](#)

[Results](#)

[LP Solution Details](#)

---

**SCOPF Solution Process**

---

The SCOPF involves three major steps that can be solved either automatically or manually from the SCOPF control dialog:

- Initialization to setup the SCOPF LP tableau and the control structures
- Contingency analysis calculation and storage of control sensitivities associated with each contingency violation
- SCOPF iterations, which include an LP solution and a power flow solution. During each LP step in the LP routine, the algorithm enforces the newest most severe contingency violation. After each violation is processed, all of the unprocessed violations are updated. This step is crucial since often resolving the most severe violation resolves numerous other violations. For instance, a single line might be overloaded in a number of contingencies: fixing the worst contingency fixes the others as well. On the other hand, processing some violations may result in new violations. In order to verify that no new violations have been created by the control changes made, the SCOPF will go back to step two and reprocess all the contingencies and the new base solution. The number of times this iteration occurs is determined by the Maximum Number of Outer Loop Iterations.

The SCOPF terminates when all the contingency violations have been processed. Note that the user can rerun the SCOPF by repeating the solution process if they want to verify the contingency violation enforcement at the new optimal operating point.

---

## SCOPF Results

---

The Results page of the [Security Constrained Optimal Power Flow](#) form provides information on the results of the latest SCOPF solution, including the [contingency violations](#) included, the [marginal price details](#), and the [marginal control details](#).



---

## SCOPF Equality and Inequality Constraints

---

Two general types of constraints are involved in the SCOPF solution: equality and inequality constraints. Equality constraints are constraints that have to be enforced. That is, they are always "binding". For example in the SCOPF, as well as in the OPF and in the power flow, the real and reactive power balance equations at system buses must always be satisfied (at least to within a user specified tolerance). In contrast, inequality constraints may or may not be binding. For example, a line MVA flow under a certain contingency may or may not be at its limit.

The SCOPF problem is solved by iterating between a power flow solution and a contingency constrained LP solution, some of the constraints are enforced during the power flow solution and some constraints are enforced during the LP solution. The constraints enforced during the power flow are, for the most part, the constraints that are enforced during any power flow solution. These include the bus power balance equations, the generator voltage set point constraints, and the reactive power limits on the generators. What differentiates the SCOPF from a standard power flow and from the OPF are the constraints that are explicitly enforced by the LP solver. These include the following constraints:

[Equality Constraints](#)

[Inequality Constraints](#)

---

## **SCOPF Equality Constraints**

---

The SCOPF equality constraints are the same as the [OPF equality constraints](#): Area MW interchange, bus MW and Mvar power balance, Generator voltage setpoint and super area MW interchange.

---

## SCOPF Inequality Constraints

---

The following classes of inequality constraints are enforced during the SCOPF solution.

### Generator real power limits

Generator real power limits are enforced during the SCOPF LP solution.

### Generator reactive power limits

Generator reactive power limits are enforced during the SCOPF LP solution.

### Interface MW Limits

Interface MW limits are enforced during the SCOPF solution. [Interfaces](#) are used to represent the aggregate flow through a number of different devices. During the SCOPF the MW post-contingency flow through the interface is constrained to be less than or equal to a user specified percentage of its limit, provided the interface is active for enforcement. For an interface to be active for enforcement the following three conditions must be met:

- Interface enforcement must not be disabled for the case. This field can be set from either the [OPF Constraints Dialog](#) or the [OPF interfaces records](#). As default, the interface enforcement is not disabled. Note that interface flow is limited to a percent of its limit as specified by the interface's [Limit Monitoring Settings](#).
- Interface enforcement must be active for at least one of the interface's areas. Note, an interface is assumed to be in each area that contains at least one of its components. This field can be set from the [OPF Area Records display](#). Note: the default is that interface enforcement is not active, so be sure to activate this if you want these constraints enforced.
- Enforcement must be active for each individual interface. This field can be specified from the [OPF interfaces records](#) display or in the [Limit Monitoring Settings](#) dialog. The default is active.

Each interface that is active for enforcement is modeled as an inequality constraint, which may be either binding or not binding. If the constraint is not binding then it does not impact the solution. If a constraint is binding then it has an associated marginal cost of contingency enforcement. When manually solving the SCOPF one can skip a contingency violation associated to the interface by setting the Include field of the [SCOPF CTG Violations](#) dialog to **No**.

### Transmission Line and Transformer (Branch) MVA Limits

Transmission line and transformer (branch) MVA limits are enforced during the SCOPF solution. During the LP the post-contingency branch line flow is constrained to be less than or equal to a user specified percentage of its limit, provided the branch is active for enforcement. For a branch to be active for enforcement the following three conditions must be met:

- Line/Transformer enforcement must not be disabled for the case. This field can be set from either the [OPF constraints dialog](#) or the [OPF Line/Transformer Records](#) display. The default is that case line/transformer enforcement is not disabled. Also note that the branch flow is limited to a percent of its limit as specified by the branch's [Limit Monitoring Settings](#).
- Branch enforcement must be active for the branch's area. Enforcement for tie-lines must be active for either area. This field can be set from the [OPF Line/Transformer Records](#) display. The default is that branch enforcement is not active, so be sure to activate this if you want these constraints enforced.

- Enforcement must be active for each individual branch. This field can be set from the [OPF Line/Transformer Records](#) display or in the [Limit Monitoring Settings Dialog](#). The default is active.

Each branch that is active for enforcement is modeled as an inequality constraint, which may be either binding or not binding under contingency conditions. If the constraint is not binding then it does not impact the solution. If a constraint is binding then it has an associated marginal cost of enforcing the contingency constraint, which is shown on the [SCOPF Bus Marginal Price Details](#) dialog. When manually solving the SCOPF one can skip a contingency violation associated to the branch by setting the Include field of the [SCOPF CTG Violations](#) dialog to **No**.

---

## SCOPF Control

---

The control dialog allows the user to manually or automatically run a SCOPF simulation and visualize the execution of the simulation. The [SCOPF solution process](#) involves three steps: base case solution and initialization, contingency analysis, and SCOPF iterations.

### Run Full Security Constrained OPF

Press this button to run the three steps of the SCOPF solution automatically. The SCOPF will solve the base case using the selected method, will use the currently stored list of contingencies during the contingency analysis step, and will take the CA results and sensitivities to iterate in order to obtain the optimal solution that minimizes cost and enforces contingency violations.

### Options

#### Maximum Number of Outer Loop Iterations

Indicates the number of maximum outer loop iterations. The outer loop iterations determines how many times the contingency analysis will be re-run following a successful SCOPF dispatch. In this manner, Simulator will look for new violations that may occur due to the new generation dispatch.

#### Consider Binding Contingent Violations from Last SCOPF Solution

When checked, this option ensures that the contingent violations from the last SCOPF solution are included in the current SCOPF solution. This option is helpful in preventing the SCOPF from hunting between having a constraint binding in one solution, and resolving with it not binding in a later solution because it was previously remedied. This option should generally always be checked, unless the user is sure that the previous solution has no bearing on the current solution, such as having made major changes to the system since the previous solution.

#### Initialize SCOPF with Previously Binding Constraints

When checked, this option results in the SCOPF solution process starts with the exact same LP tableau from the last solution. This can make for fairly fast solutions (recognizing that the contingency analysis needs to be resolved) when the changes to the system are small. Simulator automatically uses this option when doing multiple outer loops of the SCOPF. This option allows the user to solve the outer loops (set the outer loop counter to 1) by repeatedly solving the SCOPF manually, potentially making modifications between solutions if desired.

#### Set Solution as Contingency Analysis Reference Case

Check this field to set the solution of the SCOPF as the contingency analysis reference. If the system has enough controls to remove all the contingency violations, a rerun of the contingency [analysis](#) using the SCOPF solution as the reference should report no branch violations.

#### Maximum Number of Contingency Violations Allow Per Element

Specify the maximum number of contingency violations that the SCOPF analysis should allow per element. After completing the power flow solutions for the base case and the contingencies, only this number of contingency violations per element are then passed to the linear programming algorithm. The assumption is that if you fix the 10 worst contingency violations for an element then the others will also be fixed.

#### Basecase Solution Method

Specifies whether the solution of the base case is performed using the power flow algorithm or the optimal power flow algorithm. The selection will affect the initial conditions of the system and consequently the contingency analysis results and the sensitivities used by the LP solver. Currently the SCOPF does not resolve the contingency analysis during the optimization since this is computationally expensive. See the [SCOPF solution process](#) for details.

### **Handling of Contingent Violations Due to Radial Load**

It is often common when computing a security constrained OPF to have violations occur on branches due to radial load. In these instances, there is no way to adjust controls to continue to serve the load, without overloading the serving element. Therefore you can choose how contingent violations of this type should be handled by the SCOPF. You can choose to flag them but not include them in the SCOPF, ignore them completely, or include the violations in the SCOPF. Note that if you include the violations in the SCOPF, the SCOPF algorithm will not be able to remove the violation on the element via generation dispatch. However, it may be able to do so if the load in question has a load benefit curve defined and is available for load shed dispatch in the SCOPF routine.

### **DC SCOPF Options**

The DC options given for the SCOPF revolve around the treatment of Line Outage Distribution Factors (LODF) during the DC SCOPF solution. You can choose to discard the LODFs when the SCOPF is finished, store them in memory (lost when Simulator is closed,) or in memory and in the PWB file (if saved.)

If the LODFs have been stored and you wish to clear them (they can require quite a bit of RAM depending on the number of contingencies and size of the case,) you can press the button labeled **Clear Stored Contingency Analysis LODFs**.

### **SCOPF Results Summary**

#### **Number of Outer Loop Iterations**

The number of outer loop iterations required to solve the SCOPF.

#### **Number of Contingent Violations**

Number of violations from the contingency analysis portion of the solution, which are used to attempt to determine the security constrained dispatch.

#### **SCOPF Start Time, SCOPF End Time**

Physical time when the SCOPF solution process started and finished.

#### **Total Solution Time (Seconds)**

Length of time needed to determine the SCOPF solution.

#### **Total LP Iterations**

Total number of Linear Programming iterations necessary to determine the SCOPF solution.

### **Contingency Analysis Input**

#### **Number of Active Contingencies**

The number of contingencies included in the SCOPF simulation. Specific contingencies may be excluded from the simulation in the [contingency analysis dialog](#) by changing the skip field of a contingency to YES. The contingency analysis dialog can be conveniently accessed from the SCOPF control dialog.

#### **View Contingency Analysis Form**

Clicking this button shows the [contingency analysis dialog](#).

### **Contingency Analysis Results**

This window allows the user to monitor the details of the contingency analysis step. During contingencies, the outage actions, the solution of each contingency, and the solution of the CA run are reported.

---

## SCOPF CTG Violations

---

The contingency violations page is available from the Results tab of the [Security Constrained Optimal Power Flow](#) form. The display lists the results from the latest contingency analysis run including the violations that were included in the SCOPF and the final error for each violation. This dialog may change if the user reruns the SCOPF by solving the contingency analysis using the SCOPF solution as contingency analysis reference. Right click any where in the display to copy a portion or all of the display to the Window's clipboard, or to print the results.

The display shows the following fields:

### Contingency Name

This is the contingency label. By default single line contingencies start with an "L", single generator outages with a "G", and single transformer outages start with an "X".

### Category

Currently, the SCOPF considers only branch and interface violations. Thus, the category of the contingencies should be branch MVA or Interface.

### Element

Shows information about the specific element that presented the violation. When the violation occurs in a branch, this column includes the identifiers of the sending and receiving ends of the branch, the circuit, and the direction of the violating flow. Since Interfaces are directed, this field will present only the interface name in the case of violating interfaces.

### Value

The percentage flow that appears in the branch during the contingency prior to optimization. If this number is larger than the scaled limit, the violation needs to be removed.

### Scaled Limit

The scaled limit corresponds to the Line/Transformer Percentage specified in the [limit monitoring settings](#) dialog. By specifying this limit to be higher than 100% some of the contingency violations might be effectively relaxed. Sometimes this helps the OPF and the SCOPF obtain a feasible solution. On the other hand, it is often required to analyze the performance of the system if branches would have higher ratings.

### New Value

The percentage flow that appears in the branch during the contingency after SCOPF optimization. If this number is larger than the scaled limit, the contingency violation has not been removed and it is therefore unenforceable. If the value is equal to the scaled limit, then the contingency violation constraint would be binding. If the value is smaller than the scaled limit, the contingency violation has been removed.

### Error

The difference between the new value and the scaled limit. If the error is positive, the line is unenforceable. If the error is zero, the constraint has been corrected.

### Included

Indicates if the contingency violation was included as a constraint in the SCOPF solution.

### Marginal Cost

Indicates the cost associated with the contingency violation. If the constraint is unenforceable, the marginal cost is assigned arbitrarily as a high value in the [OPF constraint options](#) dialog.



**Unenforceable**

Indicates whether the contingency violations is unenforceable, i.e., the system has not enough controls to relieve the branch overload when the contingency occurs.

**Skip Violation?**

Change this field to NO if the contingency violations should not be included as a SCOPF constraint. This is sometimes useful in order to analyze the effect of the contingency violation in the SCOPF solution. This field may be toggled when doing a manual SCOPF solution.

---

## SCOPF LP Solution Details

---

The LP Solution Details page of the [Security Constrained Optimal Power Flow](#) form provides information on the linear program solution of the SCOPF, including a list of [All LP Variables](#), [LP Basic Variables](#), and [LP Basis Matrix](#). This information applies to the linear programming tableau solution method.

---

## SCOPF All LP Variables

---

The SCOPF All LP Variables dialog displays the basic and non-basic variables associated with the final LP SCOPF solution. Users interested in the specifics of the LP SCOPF can access this page to obtain internal information about the SCOPF solution. To see the display, open the [Security Constrained Optimal Power Flow](#) form, click on the LP Solution Details tab, and access the All LP Variables page.

Right click any where in the display to copy a portion or all of the display to the Window's clipboard, or to print the results.

The display lists each of the LP variables with the following fields:

### **ID**

Variable identifier.

### **Org. Value**

The initial value of the LP variable before SCOPF optimization.

### **Value**

The final value of the LP variable after SCOPF optimization.

### **Delta Value**

The difference between the original value field and the value field.

### **Basic Var**

Shows the index of the basic variables in the LP basis. If the value is zero, the variable is non-basic. These values are set up after the SCOPF calculates the contingency violation sensitivities.

### **NonBasicVar**

Shows the index of the non-basic variable. If the value is zero, the variable is basic.

### **Cost(Down)**

The cost associated with decreasing the LP variable. The field will show if the variable is at its max or min limit.

### **Cost(Up)**

The cost associated with increasing the LP variable. The field will show if the variable is at its max or min limit.

### **Down Range**

The available range to decrease the basic variable before a new constraint is hit under a contingency condition.

### **Up Range**

The available range to increase the basic variable before a new constraint is hit under a contingency condition.

### **Reduced Cost Up**

The cost reduction that would be experimented if a LP variable increases. If a constraint is at the limit, the field shows the change in cost of constraint enforcement.

### **Reduced Cost Down**

The cost reduction that would be experimented if a LP variable decreases. If a constraint is at the limit, the field shows the change in cost of constraint enforcement.

**At Breakpoint**

Yes, if the LP variable is at a break point.

---

**SCOPF LP Basic Variables**

---

The LP Basic Variables (from the SCOPF [LP Solution Details](#) page) displays the basic variables of the final LP solution. The basic variables may correspond to controls that can be altered to minimize the objective function, or slack variables associated with unenforceable constraints. Users interested in the specifics of the LP SCOPF can access this page to obtain internal information about the SCOPF solution. Right click any where in the display to copy a portion or all of the display to the Window's clipboard, or to print the results.

The display lists each LP variable with the following fields:

**ID**

Basic variable identifier.

**Org. Value**

The initial value of the basic LP variable before the SCOPF optimization.

**Value**

The final value of the basic LP variable after the SCOPF optimization.

**Delta Value**

The difference between the original value field and the value of the basic variable.

**Basic Var**

Shows the indices of the basic variables in the LP basis.

**Cost(Up)**

The cost associated with increasing the basic variable.

**Down Range**

The available range to decrease the basic variable before a new constraint is hit under a contingency condition.

**Up Range**

The available range to increase the basic variable before a new constraint is hit under a contingency condition.

---

**SCOPF LP Basis Matrix**

---

The LP Basis Matrix page of the SCOPF [LP Solution Details](#) tab displays the basis matrix associated with the final SCOPF LP solution. There is one row per constraint and one column per basic variable. Additional columns summarize information associated with each constraint. This page is usually only of interest to users interested in knowing the specifics of the SCOPF solution. Knowing the basis matrix can be helpful in figuring out why a particular SCOPF solution exhibits a certain behavior. The entries in the basis matrix give the sensitivity of each constraint to each of the basic variables.

As any case info display in simulator, right click to see options to copy information to the clipboard and to perform standard windows actions, such as printing.

---

**SCOPF Bus Marginal Price Details**

---

This display (from the SCOPF [Results](#) page) shows information about the components of the marginal cost at each bus. The display is relevant to see how the bus marginal cost depends on the cost of enforcing system constraints such as branch limits and area equality constraints. This display is useful for indicating which constraints are contributing towards the determination of the marginal price at each bus.

As any case info display in Simulator this display can be customized and the information copied, printed, and saved by accessing the local menu option with the mouse right click.

One key feature of this display is the breakdown in the bus MW marginal cost into three components: the cost of energy, the cost of congestion, and the cost of losses. Also, while the bus MW marginal cost does not change based on the reference, the costs of energy, congestion, and losses are reference dependent. This reference is set on a per-area (or per-super area) basis in the [Area information dialog OPF tab](#) (or the [Super Area information dialog OPF tab](#)).

---

## SCOPF Bus Marginal Controls

---

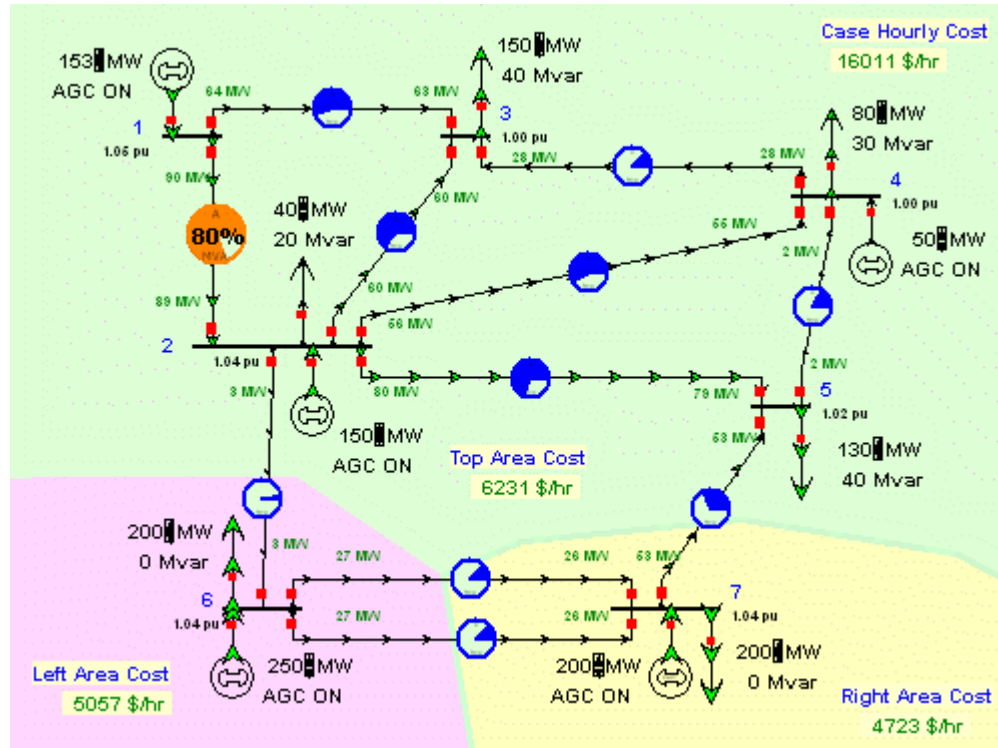
This display (from the SCOPF [Results](#) page) shows the sensitivities of the controls with respect to the cost at each bus. A change in a system control will have the indicated effect in the marginal cost at the system buses. Vice-versa, the marginal cost at a bus is affected by changes in the value of the basic variables.

The Marginal Controls page can be accessed by opening the [Security Constrained Optimal Power Flow](#) form, and clicking on the Results tab.



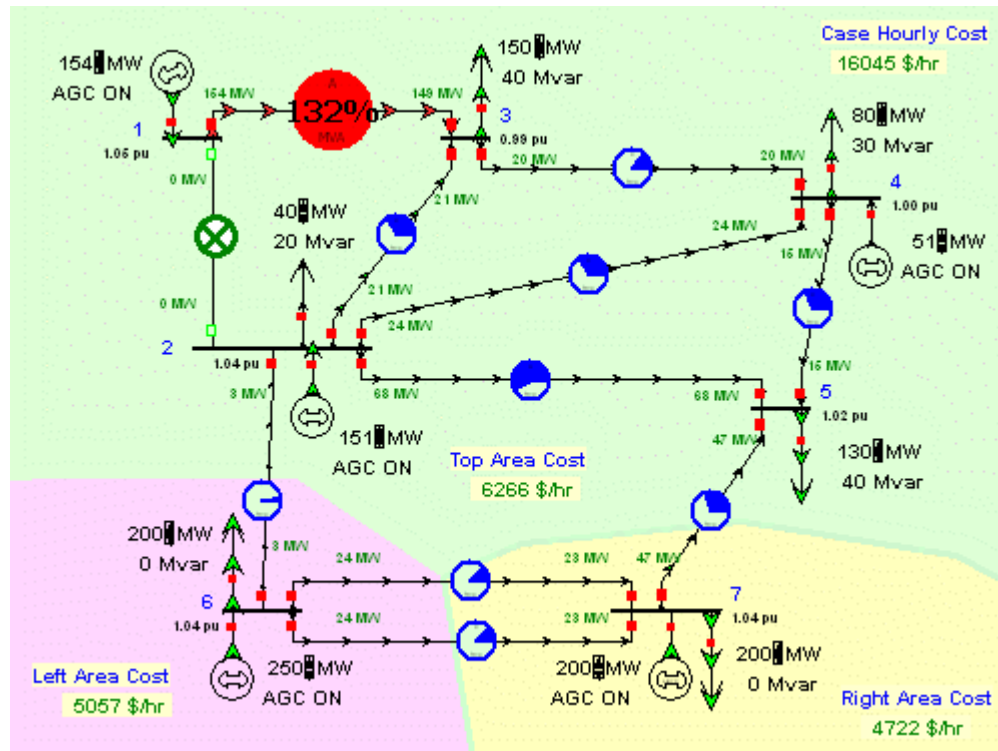
## SCOPF Example: Introduction

In this section we introduce an example of using the SCOPF. Consider the seven bus, three area system contained in the file B7SCOPF (included with the PowerWorld Simulator). This case is the same case used in the OPF example, except that the line from bus 2 to 3 has now a 100MVA rating, lines 1 to 2, 1 to 3 and 2 to 5 have a 120MVA rating, and all three areas are initially on OPF control. To initially solve the case using the optimal power flow, select **Primal LP** in the **OPF** ribbon group on the **Add Ons** ribbon tab. The solution obtained is shown below. The total case hourly cost is \$16,011 / hr.



B7SCOPF Case Solved using OPF

We are interested in determining an optimal solution that meets security constraints under contingency conditions. In order to show that the current OPF solution does not enforce contingency violations, take line 1 to 2 out of service by clicking in a circuit breaker. Then solve the power flow by pressing the **Single Solution** button. The result indicates that line 1 to 2 is overloaded 32%. The SCOPF algorithm will attempt to move the operating solution such that no contingency violation occurs in the system. Close line 1 to 2 back in service.

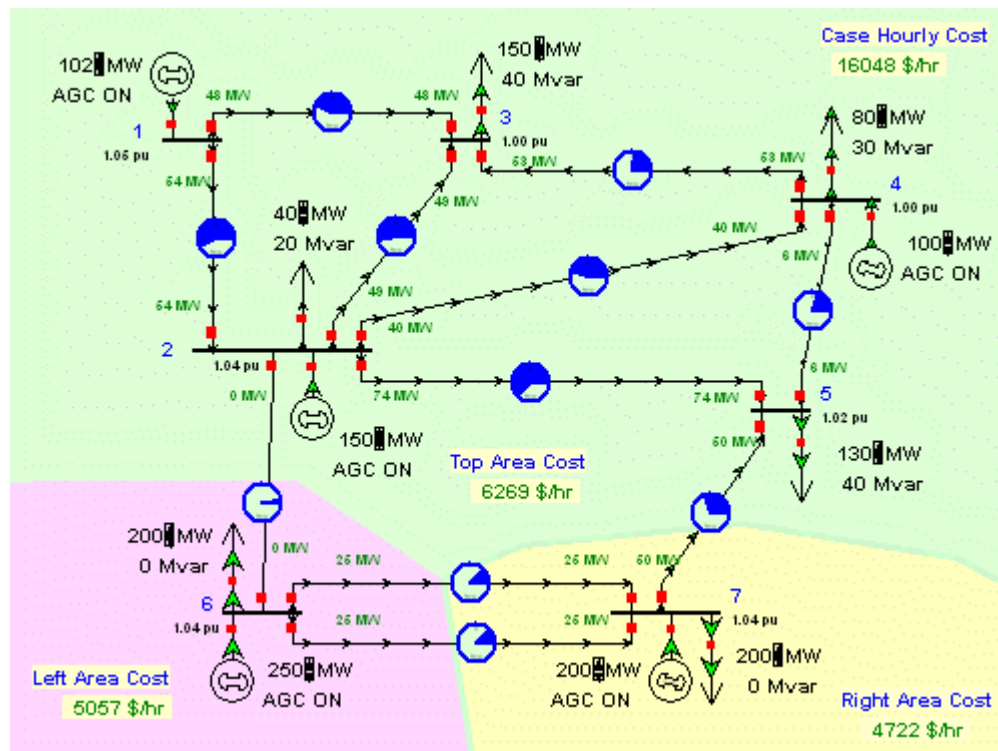


B7SCOPF Power Flow Solution with line 1-2 open

The next step is to specify the contingency conditions that the system should withstand. In order to do that, we access the [Contingency Analysis dialog](#) under **Tools**. Note that we can also access this dialog from the [SCOPF control dialog](#) by selecting **Add Ons > Security Constrained OPF** and pressing the **View Contingency Analysis Form** button on the **Options** page. The B7SCOPF case does not have contingencies associated with it. Insert single line contingencies in the contingency list by pressing the **Auto Insert** button located at the bottom left of the Contingency Analysis dialog. In the [Auto Insert Dialog](#) select the option for **Single Transmission Line or Transformer** and select **Numbers** under the *Identify buses by* field. Select **Do Insert Contingencies**. This will prompt to insert 11 single line contingencies corresponding to all the lines in the system. Select **Yes**. You can now close the Contingency Analysis dialog.

Return to the **Security Constrained Optimal Power Flow Form**. You can set the SCOPF to use the OPF solution as the base by selecting **Solve base case using optimal power flow**.

You can now solve the SCOPF by pressing the **Run Full Security Constrained OPF**. This will process the contingency violations and iteratively solve the LP program and the power flow equations to minimize the objective function and enforce equality and inequality constraints. The solution is shown below. The total operating cost is now \$16,048 /hr. The increase in operating cost is due to enforcing security constraints. If a new contingency analysis is performed using the optimal solution as the reference, it will be found that no contingency violations occur for the contingencies in the list, i.e., branch flows are less than (or equal to) 100% in the post contingency condition. Thus, the system meets all the specified constraints. You can analyze the SCOPF results by browsing the information in the SCOPF tabs. Note that the **CTG Constraints** dialog does not show unenforceable constraints, but the branch violation of line 2 to 5 due to the contingency 5 to 7 is now binding.



B7SCOPF Case Solved using SCOPF: The system now meets the contingency constraints.

## SCOPF Example: Marginal Prices

Using the SCOPF solution from the previous page, select the **Bus Marginal Price Details** from the **Results** page of the **SCOPF Dialog** to view the detail of the marginal price components. Note in the following Figure that each area constraint contributes equally to the marginal cost of the buses in that area. The binding inequality constraint from bus 2 to 5 makes further contribution to the bus marginal price of buses in area Top.

The screenshot shows the 'Security Constrained Optimal Power Flow Form' window. The 'SCOPF Status' is 'SCOPF Solved Correctly'. The 'Options' tab is selected, and the 'Bus Marginal Price Details' sub-tab is active. The table below displays the marginal price components for seven buses.

	Number	Name	Area Name	MW Marg. Cost	Energy \$/MWh	Congestion \$/MWh	Losses \$/MWh	Area 1 MW Constraint	Area 2 MW Constraint	Area 3 MW Constraint	Line from 1 to 3 ckt. 1
1	1	1	Top	13.00	12.85	0.15	0.00	12.85	0.00	0.00	0.00
2	2	2	Top	12.85	12.85	0.00	0.00	12.85	0.00	0.00	0.00
3	3	3	Top	13.75	12.85	0.90	0.00	12.85	0.00	0.00	0.00
4	4	4	Top	14.00	12.85	1.15	0.00	12.85	0.00	0.00	0.00
5	5	5	Top	17.82	12.85	4.96	0.00	12.85	0.00	0.00	0.00
6	6	6	Left	17.60	17.60	0.00	0.00	0.00	17.60	0.00	0.00
7	7	7	Right	21.80	21.80	0.00	0.00	0.00	0.00	21.80	0.00

Seven Bus Case SCOPF Bus Marginal Price Details

## SCOPF Example: Unenforceable Constraints

Using the same example as above, reduce the MVA rating of line 2 to 5 to 90 MVA. The initial base case LP OPF solution is the same as in the previous example. Consider the same contingency list. When the contingencies are solved using the initial OPF solution as the reference, seven contingency violations need to be removed. The CTG dialog after simulation is presented in the following figure.

	Contingency Name ▲	Element	Value	Scaled Limit	New Value	Error	Included	Marginal Cost	Unenforceable
1	L_00001-00002C1	Br 1 ( 1 ) -> 3 ( 3 ) CKT 1 at 1	157.31	120.00	-2.76	0.00	NO	0.00	NO
2	L_00001-00003C1	Br 1 ( 1 ) -> 2 ( 2 ) CKT 1 at 2	154.45	120.00	10.45	0.00	NO	0.00	NO
3	L_00002-00003C1	Br 2 ( 2 ) -> 5 ( 5 ) CKT 1 at 2	90.73	90.00	68.68	0.00	NO	0.00	NO
4	L_00002-00004C1	Br 2 ( 2 ) -> 5 ( 5 ) CKT 1 at 2	92.52	90.00	66.18	0.00	NO	0.00	NO
5	L_00006-00007C1	Br 2 ( 2 ) -> 5 ( 5 ) CKT 1 at 2	91.60	90.00	73.24	0.00	NO	0.00	NO
6	L_00006-00007C2	Br 2 ( 2 ) -> 5 ( 5 ) CKT 1 at 2	91.60	90.00	73.24	0.00	NO	0.00	NO
7	L_00007-00005C1	Br 2 ( 2 ) -> 5 ( 5 ) CKT 1 at 2	127.56	90.00	104.48	14.48	YES	1000.00	YES

CTG Dialog after SCOPF Solution

We note that the constraint from bus 2 to 5 under a contingency from 5 to 7 is unenforceable. There are not enough system controls to enforce the contingency constraint. A \$ 1,000 / hr cost is assigned to unenforceable constraints in this case. The cost of not enforcing constraints can be specified in the [OPF Constraint Options](#) Dialog.

---

## Available Transfer Capability (ATC) Analysis

---

The ATC Analysis tool is only available if you have purchased the ATC add-on to the base Simulator package. [Contact PowerWorld Corporation](#) for details about ordering the ATC version of Simulator.

Available Transfer Capability (ATC) analysis determines the maximum incremental MW transfer possible between two parts of a power system without violating any specified limits. Most often, this transfer is between two areas in the system, but can be customized to specific groups of system devices.

Simulator's ATC analysis makes use of several tools that are available elsewhere in Simulator. These include:

- [Power Transfer Distribution Factors \(PTDFs\)](#): determine the linear impact of a transfer (or changes in power injection) on the elements of the power system.
- [Line Outage Distribution Factors \(LODFs\)](#): determine the linear impact of a line outage on the elements of the power system.
- [Contingency Analysis](#): studies the impact of a list of contingencies on the power system.
- [Limit Monitoring Settings](#): control which elements of the system are monitored for limit violations.

You do not directly use these other tools when using Simulator's ATC analysis tool, but Simulator uses the settings and algorithms in the background to determine ATC. Thus, it is helpful to be knowledgeable on their use, as it will help you in interpreting the results of an ATC analysis.

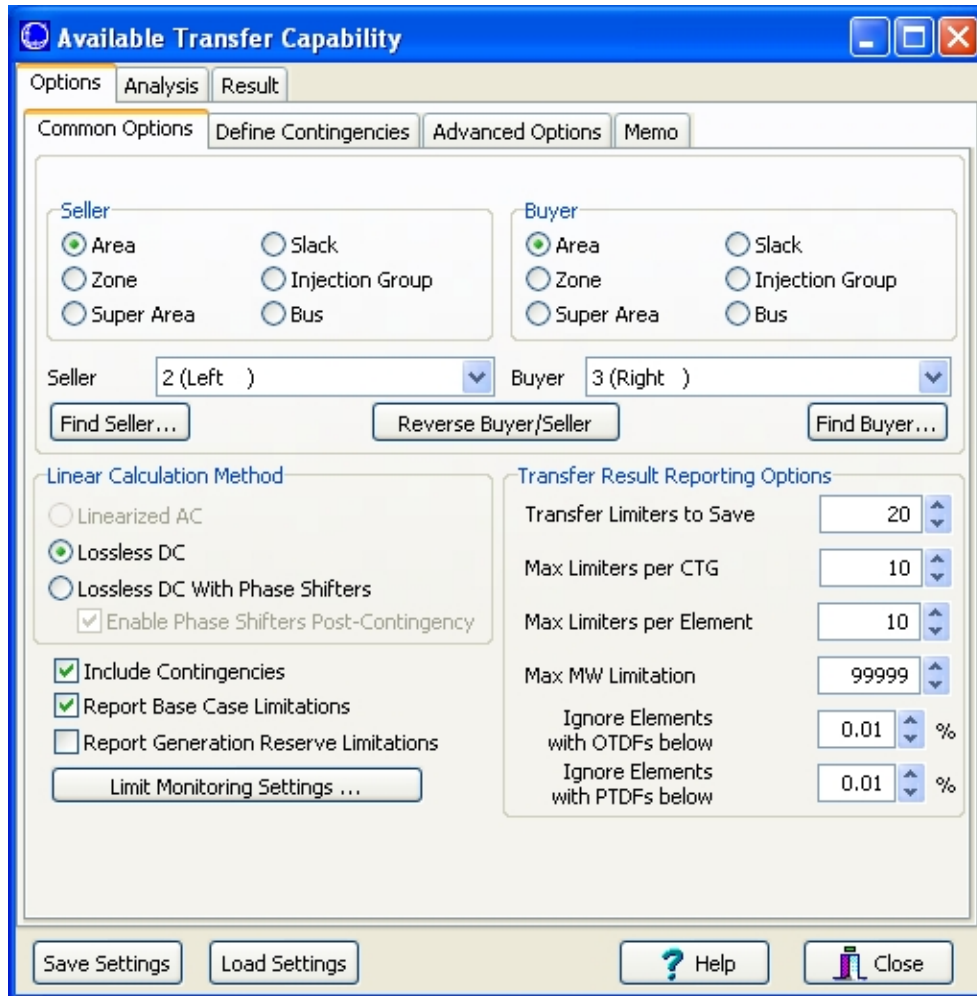
- Simulator provides three methods of determining the ATC for a transfer direction. See : [ATC Solution Methods](#)

For information on how to use the Simulator ATC tool, see [Available Transfer Capability Dialog](#).

## Available Transfer Capability Dialog

The Available Transfer Capability Dialog provides an interface for performing and viewing the results of [Available Transfer Capability Analysis](#). ATC analysis is typically only done on the present power system state or scenario (called Single Scenario ATC Analysis). Click [here](#) for information on performing [multiple scenarios](#) analysis.

To open the ATC Dialog, go to the [Add Ons](#) ribbon tab and select **Available Transfer Capability (ATC)** from the **ATC** ribbon tab. The dialog opens to the Single Scenario ATC Analysis version with the Options tab, Common Options sub-tab visible.



ATC Dialog

The ATC Dialog is divided in three tabs: [Options tab](#), [Analysis tab](#), and [Result tab](#). If Analyze Multiple Scenarios has been checked on the Advanced Options sub-tab of the [Options tab](#), the layout of the ATC Dialog is slightly different—see [Multiple Scenarios ATC Dialog](#).

In addition, the following buttons can be used to save/load ATC analysis settings:

### Save/Load Settings

ATC Analysis settings can be saved by selecting **Save Settings** on the ATC dialog. This allows you to repeat the analysis without having to reconfigure the settings. Select **Load Settings** to retrieve previously saved settings.



## ATC Dialog: Options Tab

The Options Tab of the ATC Dialog is subdivided in four sub tabs: Common Options, Define Contingencies, Advanced Options, and Memo. They have the following controls:

### Common Options

The screenshot shows the 'Available Transfer Capability' dialog box with the 'Options' tab selected. The 'Common Options' sub-tab is active. The 'Seller' section has radio buttons for 'Area' (selected), 'Zone', 'Super Area', 'Slack', 'Injection Group', and 'Bus'. The 'Buyer' section has similar radio buttons, with 'Area' selected. Below these are dropdown menus for 'Seller' (set to '2 (Left)') and 'Buyer' (set to '3 (Right)'), along with 'Find Seller...', 'Reverse Buyer/Seller', and 'Find Buyer...' buttons. The 'Linear Calculation Method' section has radio buttons for 'Linearized AC', 'Lossless DC' (selected), and 'Lossless DC With Phase Shifters', with a checked checkbox for 'Enable Phase Shifters Post-Contingency'. There are also checkboxes for 'Include Contingencies', 'Report Base Case Limitations', and 'Report Generation Reserve Limitations', and a 'Limit Monitoring Settings ...' button. The 'Transfer Result Reporting Options' section includes numeric input fields for 'Transfer Limiters to Save' (20), 'Max Limiters per CTG' (5), 'Max Limiters per Element' (5), and 'Max MW Limitation' (99999), along with percentage inputs for 'Ignore Elements with OTDFs below' (0.50%) and 'Ignore Elements with PTDFs below' (0.50%). At the bottom are 'Save Settings', 'Load Settings', 'Help', and 'Close' buttons.

### Seller Type, Buyer Type

For the [ATC Solution Method of Single Linear Step \(SL\)](#), ATC can be calculated for transfers between combinations of areas, zones, [super areas](#), [injection groups](#), buses, or to the system slack bus. For the iterated linear step ATC Solution Methods ([IL](#) and [IL then full contingency solution](#)), ATC can be calculated for transfers between areas, [super areas](#) or [injection groups](#) only. Use the seller type and buyer type options to indicate the type of the selling and purchasing entities.

### Seller, Buyer

These dropdown boxes allow you to select the selling and buying entities. The contents of each depend on the Seller/Buyer Types selected. Clicking the **Find Seller** and **Find Buyer** buttons allows you to use Simulator's [Advanced Search Engine](#) to locate the desired entities.

The Seller defines where power is injection into the system and the Buyer defines where power is taken out of the system. The transfer is simulated by increasing generation or

decreasing load in the Seller and decreasing generation or increasing load in the Buyer. The [ATC Analysis Methods - Solution Methods](#) go into greater detail about how the impact of the transfer is actually determined. When areas, zones, or super areas are selected to be either the Seller or Buyer, only on-line generators in the defined group that are on AGC control are allowed to participate in the transfer. Each generator participates in the transfer in proportion to its participation factor. When injection groups are selected to be either the Seller or Buyer, only on-line generators and loads defined with the injection group are allowed to participate in the transfer. Each element participates in the transfer in proportion to the participation factor defined with its [participation point](#) in the injection group. When a single bus is selected as the Seller or Buyer then all of the power injection change needed for the transfer comes from that bus. It does not matter if there is a generator or load at that bus. When the slack is selected for the Seller, then all of the power injection change needed for the seller side of the transfer comes from the island slack bus for the island in which the Buyer belongs. When the slack is selected as the Buyer, then all of the power injection change needed for the buyer side of the transfer comes from the island slack bus for the island in which the Seller belongs.

Generator and load MW limits are not enforced regardless of the type of buyer and seller. Error messages will be displayed if the Seller and Buyer are not both completely contained within the same electrical island.

#### **Reverse Buyer/Seller Button**

Click this button to reverse the direction currently shown. The buyer becomes the seller, and the seller the buyer.

#### **Linear Calculation Method**

The ATC analysis tool can use either a **Lossless DC** or **Lossless DC with Phase Shifters** calculation method for obtaining the ATC results.

If you select the Lossless DC option, branch flow sensitivity is calculated by estimating the real power that flows through the monitored element only from the difference in angles measured across its terminals.

The Lossless DC with Phase Shifters method, a modification to the lossless dc approximation, takes into account phase shifter operation. It is especially useful when the ATC tool continually reports overloads on branches that obviously will not overload because of the operation of a phase shifting transformer. This method assumes that the phase shifter angles may change to any value to ensure that the line flow on those lines does not change.

The Linearized AC method is not yet available.

#### **Enable Phase Shifters Post-Contingency**

This option becomes enabled when you choose the **Lossless DC with Phase Shifters** linear calculation method. It allows you to choose whether or not phase shifter control should be enforced during post-contingency ATC calculations. When this option is checked, it is assumed that phase shifter angles may change to keep the line flow from changing after the contingency has been applied. Otherwise, it is assumed that phase shifters only maintain the flow in the pre-contingency state and the angles do not change in the post-contingency state. Base case (pre-contingency) transfer limiters will still be determined with phase shifters enforced, regardless of the setting of this option when the Lossless DC with Phase Shifters option is checked.

#### **Include Contingencies Check Box**

Check this box to include contingencies (inserted or loaded using the [Contingency Analysis Tool](#)) in the ATC analysis. Note that minimizing the number of contingencies considered

greatly improves solution speed as well as computer memory requirements for doing ATC Analysis. Therefore, be careful in choosing which contingencies to use with the ATC tool.

The ATC tool uses the same contingency records as the contingency analysis tool. Contingency results may be invalid following ATC analysis. If necessary, save contingency results to an auxiliary file prior to running the ATC analysis.

#### **Report Base Case Limitations Check Box**

When checked, the ATC tool will report transfer limitations from the base case (pre-contingency).

#### **Report Generation Reserve Limitations Check Box**

When checked, the ATC tool will report transfer limitations from generation reserve.

Generator and load MW limits are not enforced when calculating linear sensitivities used to determine the transfer MW limits. However, checking this option provides an indication of when power transfers are limited based on generator maximum or minimum MW limits or the amount of load that is on-line. When this option is checked and there are limitations to how much power can be transferred because of max/min generator limits or the on-line load, additional Transfer Limiters will be present in the [results](#). The Limiting Element will indicate either the Buyer or Seller and the Transfer Limit in MW will be the amount of transfer that can be achieved before the Buyer or Seller exceeds some limitation to generation or load. When considering reserve limits for the Seller, generators will be included in the reserve limit by taking the difference between the generator maximum MW limit and the present output and loads will be included in the reserve limit by the present amount of on-line load. When considering reserve limits for the Buyer, generators will be included in the reserve limit by taking the difference between the present output of the generators and the minimum MW limit and loads will be included by an arbitrary value of 9999.9 MW. Loads are included in this manner because maximum and minimum limits on loads are not currently used in the ATC tool.

For example, assume than an area that currently has 200 MW on-line and a total maximum MW output of 250 MW for all on-line generators that are on AGC control is defined as the Seller. A 50 MW generation reserve limitation will be reported for this area. As another example, assume that an injection group currently has 50 MW of on-line generation with a total maximum MW output of 150 MW and 50 MW of on-line load is defined as the Seller. A 150 MW reserve limitation will be reported for this injection group.

No generation reserve limitations are reported when the seller or buyer type is a bus or the slack.

If using one of the iterated methods, this option will be ignored except during the first single linear step. Reserve limitations may be reported from this step, but limitations of this type will not be iterated on during the individual iteration process. If choosing to enforce generator MW limits appropriately with the type of transactors that are defined, reserve limits will still be reported, regardless of this option setting, if they are hit during any step of the iterative process when ramping is occurring. See the detailed descriptions of either the [Iterated Linear Step \(IL\)](#) or [Iterated Linear Step \(IL\) then Full CTG Solution](#) method for more information about how reserve limits are checked during the iterated methods.

#### **Limit Monitoring Settings Button**

Click this button to open the [Limit Monitoring Settings Dialog](#). Limiting Elements will only be reported for those elements that are selected to be monitored. Note that minimizing the number of monitored power system elements greatly improves solution speed as well as computer memory requirements for doing ATC Analysis.

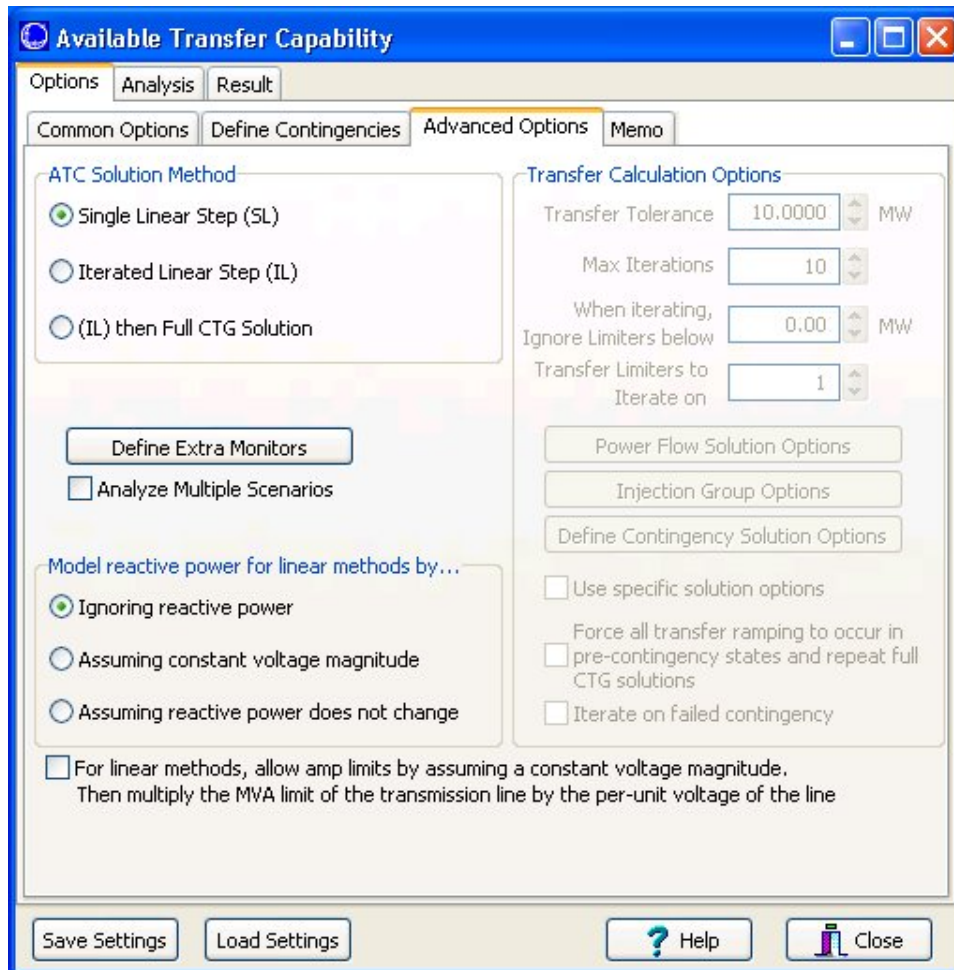
#### **Transfer Result Reporting Options**

The options in this section of the ATC dialog are discussed in detail in the topic on [Transfer Result Reporting Options](#).

### Define Contingencies

The Define Contingencies tab is similar to the [Contingencies Tab](#) of the [Contingency Analysis Dialog](#). The user can insert, auto-insert, define and/or delete contingency records. See [Contingency Analysis](#) for detailed information on defining contingencies.

### Advanced Options



### ATC Solution Method

One of the [solution methods](#) for determining the ATC of a specified transfer direction.

### Define Extra Monitors

Simulator's ATC tool determines the maximum amount of MW transfer between the seller and buyer. If you would like to also determine the flow on additional lines or interfaces at the transfer levels determined by the ATC tool, you can utilize Extra Monitors. Click the **Define Extra Monitors** button to open the [ATC Extra Monitors Dialog](#). The flow on the extra monitors will be reported for each transfer limiter determined.

### Analyze Multiple Scenarios

Check Analyze Multiple Scenarios to perform ATC Analysis on several scenarios. See [ATC Dialog for Multiple Scenarios](#) for more information on Multiple Scenario Analysis.

### Model Reactive Power for Linear Methods By...

The linearized methods used in the ATC tool are based only on the changes in real power MW in the system, thus an assumption needs to be made about the reaction of the Mvar flows during the linear calculations. The chosen assumption allows a MW limit to be calculated from the specified MVA limit for use in determining the ATC values. The choices are:

- **Ignoring reactive power** – reactive power is completely ignored. The MW limit of a line is assumed to be the MVA limit of the line.
- **Assuming constant voltage magnitude** – reactive power is adjusted to hold voltage magnitudes constant
- **Assuming reactive power does not change** – reactive power is held at pre-ATC solution levels. The MW limit for each line is calculated from the MVA limit and the reactive power at the pre-ATC solution level.

### For Linear Methods, Allow Amp Limits by Assuming a Constant Voltage Magnitude

If checked, Simulator will allow converting MVA limits to Amp limits by assuming constant voltage magnitudes based on the base case full AC load flow operating point just prior to the ATC linear calculations. This option will only be used if the [Limit Monitoring Settings Option of Treat Line Limits as Equivalent Amps](#) is checked.

This option will cause the ATC tool to internally use a modified MVA limit which is equal to the user-specified MVA rating multiplied by the actual terminal voltages of the line. This results in a modified MVA rating which is higher than the user-specified MVA rating when the voltage on the line is higher than nominal and lower when the voltage is lower than nominal. For purposes of calculating the ATC values, a MW limit is determined from the MVA rating based on how this option is set and how the reactive power is modeled.

### Transfer Calculation Options

**Note:** The Transfer Calculation Options section is disabled if the [Single Linear Step](#) Solution Method is selected.

#### Transfer Tolerance

This is the tolerance used during the [Iterated Linear Step](#) or [\(IL\) then Full CTG Solution](#) methods. The iterated methods will finish if the transfer step amount for the next iteration is less than this tolerance. The default is 10 MW.

#### Max. Iterations

This is the maximum number of iterations used in the [Iterated Linear Step](#) or [\(IL\) then Full CTG Solution](#) methods. The default is 10.

### When Iterating, Ignore Limiters below

The iterated techniques perform additional analysis on individual transfer limitations in the order of increasing Transfer Limit in MW. When using one of the iterated techniques, transfer limitations with Transfer Limit values below a certain threshold can be ignored for the iterated portion of the process. Transfer limitations with Transfer Limit values equal to or below the value set with this option may still appear in the results, but they will not be iterated on individually.

Once an individual limiter is being iterated on, this value will no longer be used. This means that during the individual iteration process, the transfer level for a limiter can fall below this threshold.

### Transfer Limiters to Iterate on



This is the number of transfer limiters to iterate on in the [Iterated Linear Step](#) or [\(IL\) then Full CTG Solution](#) methods. The default is 1. The transfer limiters actually iterated on are determined by selecting the number of limiters set in this option in ascending order of Transfer Limit with the Transfer Limit at or above the value set in the **When Iterating, Ignore Limiters below** option.

#### Power Flow Solution Options

This button brings up the [Power Flow Solution Options Dialog](#).

#### Injection Group Options

This button brings up a dialog from which options associated with injection groups can be set. These are the same options that get set for injection groups when using [Island-based AGC](#) and dispatching using an injection group; the ATC process switches to using an island-based AGC when iterating and using injection groups for the seller and buyer. The options that can be set are Island AGC Tolerance, Allow only AGC units to vary, Enforce unit MW limits, Do not allow negative loads, and how reactive power load should change as real power load is changed.

#### Define Contingency Solution Options

Click this button to open the [Contingency Solution Options Dialog](#).

#### Use Specific Solution Options for Contingencies

When checked, the ATC tool will use the solution options defined by pressing the **Define Contingency Solution Options** button for contingency analysis. When not checked, all solutions will use the options defined by pressing the **Power Flow Solution Options** button.

#### Force all transfer ramping to occur in pre-contingency states and repeat full CTG solutions

When using the iterated solution methods, the typical iterated operation is to solve for the transfer step amount in the post-contingency state, and then apply the transfer step amount to the post-contingency state and resolve for the next transfer step amount. When this option is checked, the transfer step amount for each iteration is calculated in the post-contingency state, but the power flow state is restored to the pre-contingency state before the transfer step amount is applied. The contingency is then applied after the transfer step amount has been ramped and the transfer step amount for the next iteration is calculated after the contingency has been applied. The purpose of this is to take into account the possibility of conditional actions in the contingency definitions that may not be active in the early steps of the iterated calculations, but may become active at some point during the application of the iterated steps.

#### Iterate on failed contingency

This option is only enabled when the option to **Force all transfer ramping to occur in pre-contingency states and repeat full CTG solutions** is selected. When this option is checked and the power flow fails to solve after the contingency has been applied, a new iteration loop is entered to determine the highest transfer level at which the contingency can be solved. This loop attempts to determine a more accurate transfer level at which the contingency fails to solve rather than simply reporting the transfer level at which the last power flow was successful regardless of the current step size. To start the iterative process, the current step size is reduced by half. The ramping is performed and the contingency is solved. The step size will remain the same as long as the contingency can be solved. If the contingency fails to solve, the step size will be reduced by half again. The ramping will continue until the contingency no longer solves, the accumulated transfer during the process meets the original step size, or the step size becomes smaller than the step size tolerance.

An additional check is done during the iterative process on the failed contingency. When the contingency does solve, the flow on the monitored element is checked to determine if it exceeds its limit. If it is overloaded, the step size will be reduced by half until the monitored element is no longer over its limit. While this check is being done on the monitored element limit, the ramping continues out to the specified amount and the contingency is solved.

During the entire iterative process on a failed contingency solution, the step size is limited to be between zero and the step size which was ramped when the contingency failed to solve for the first time.

### **Memo**

The Memo tab allows you to enter a note related to the ATC analysis.

---

## Transfer Result Reporting Options

---

### Transfer Limiters To Save

This value tells Simulator how many total Transfer Limiters to save. Simulator will save those Transfer Limiters with the lowest Transfer Limitation. An explanation of a Transfer Limiter follows:

During Linear ATC Analysis, Simulator determines the Transfer Limitation (See [Available Transfer Capability Analysis](#)) for each transmission line and interface during each contingency and the base case. From this Simulator develops a list of Transfer Limiters. A Transfer Limit contains three key pieces of information:

- Transfer Limit in MW
- Limiting Element: Transmission branch (or interface) that causes the limit
- Limiting Contingency: Contingency that is applied to cause the Limiting Element to overload (if it is a limit without any contingency applied, then the contingency will say *Base Case*)

Thus if we are monitoring 1000 transmission lines during 99 contingencies plus 1 base case, there would be 100,000 Transfer Limiters calculated. We are not concerned with all 100,000 limitations, therefore, only the limitations with the smallest Transfer Limit in MW are reported.

### Max Limiters per CTG

When analyzing a long list of contingencies, the worst transfer limitations may all occur during the same contingency. Set this value to limit the number of Transfer Limiters saved that are associated with one contingency.

### Max Limiters per Element

When analyzing a long list of contingencies, the worst transfer limitations may all be overloads of the same limiting element. Set this value to limit the number of Transfer Limiters saved that are associated with one Limiting Element.

### Max MW Limitation

This value defines the maximum transfer to report between the buyer and seller. Simulator will compute the ATC analysis results until the transfer amount reaches the value in this field and only report the results meeting the other reporting criteria up to this MW limitation.

When using one of the iterated methods, this option is only applied during the initial Single Linear Step portion of the method. Once individual limiters are being iterated on, this option will be ignored.

### Ignore Elements with OTDFs below

Simulator will not report Transfer Limitations for elements with [OTDF](#) values (or [PTDF](#) values if there is no Limiting Contingency) less than this user-specified value. The default value is 0.5%, meaning that for a 100 MW transfer, there would be only a 0.5 MW increase in flow on the Limiting Element.

The transfer limitation functions involve dividing by the [PTDF](#) or [OTDF](#) values for each branch or interface to calculate the Transfer Limit. This leads to two facts:

- The accuracy of the transfer limitation is less for lines that have very small [PTDF](#) or [OTDF](#) values.
- A very small [PTDF](#) or [OTDF](#) value means that the transfer has very little impact on the line.

These two facts often result in Linear ATC analysis reporting inaccurate transfer limitations for lines that are largely unaffected by the transfer. It is not uncommon to have a transfer limitation report an extremely negative transfer limit (e.g. -1.9E28 MW). A branch which is



overloaded by a very small percentage, but which has a very small OTDF value often causes this. If the OTDF value is 0.001%, then a branch overloaded by 1 MW will result in a transfer limitation of -100,000 MW.

This motivates the usefulness of ignoring elements with small [PTDFs](#) and [OTDFs](#).

**Ignore Elements with PTDFs below**

Simulator will not report Transfer Limitations for elements with [PTDF](#) values less than this user-specified value. The default value is 0.5%.

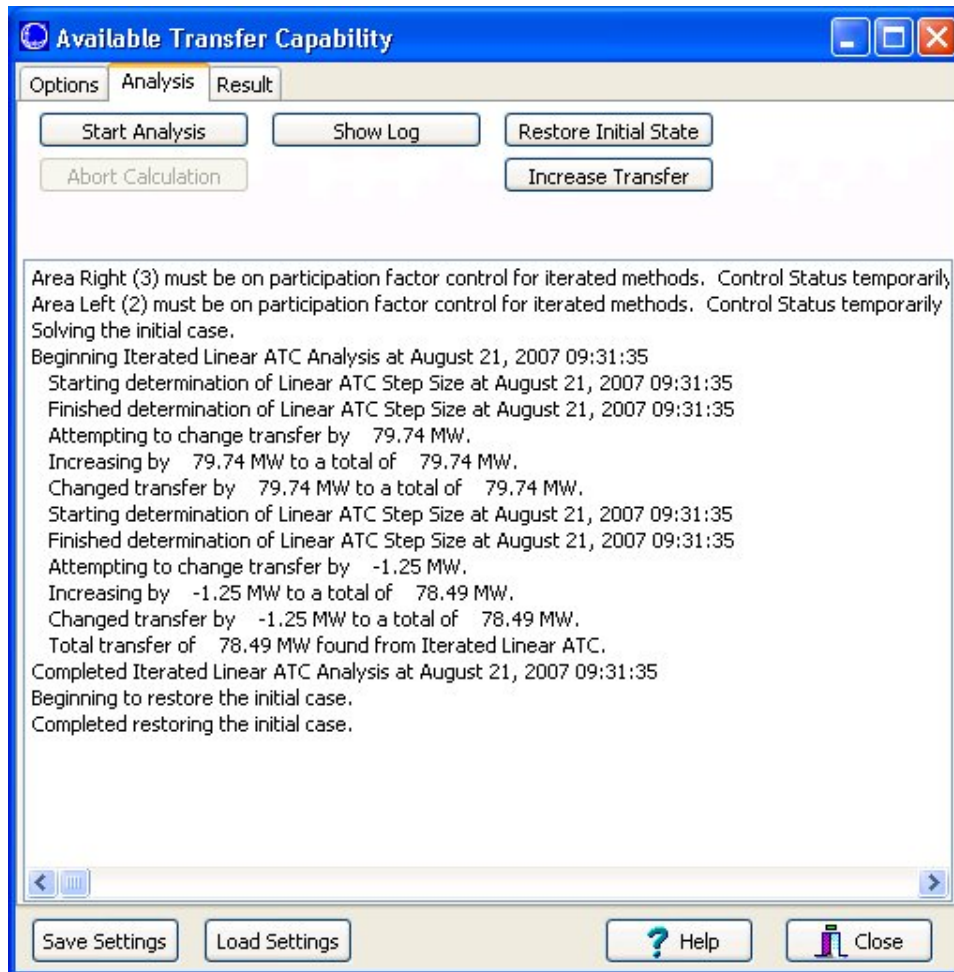
Both the PTDF and OTDF cutoff values are applied for each monitored element to determine if it should be ignored. If an element is monitored in the base case, the PTDF must meet the PTDF cutoff AND the OTDF cutoff values. If an element is being monitored for a contingency condition, the OTDF for the monitored element with the contingency in place must meet the OTDF cutoff, and the PTDF for the monitored element without the contingency in place must meet the PTDF cutoff.

---

## ATC Dialog: Analysis Tab

---

The Analysis Tab of the ATC Dialog is used to control the analysis process. The scrollable window displays a record of user-initiated actions relating to the analysis.



### Analysis Tab

The Analysis tab has the following controls:

#### Start Analysis

The **Start Analysis** button begins the ATC Analysis using the settings and options defined by the user in the ATC Analysis Dialog. Progress of the ATC Analysis is shown in the scrollable pane of the ATC Dialog – Analysis Tab window.

#### Abort Calculation

When using one of the iterative ATC Solution Methods, or while analyzing Multiple Scenarios, click this button to abort the calculation.

Note: This does not immediately abort the solution. Simulator must restore the system state before completing the abort.

**Show Log**

Click this button to show the [Power Flow Solution Message Log](#).

**Restore Initial State**

Click this button to restore the system state to the state when the dialog was first opened.

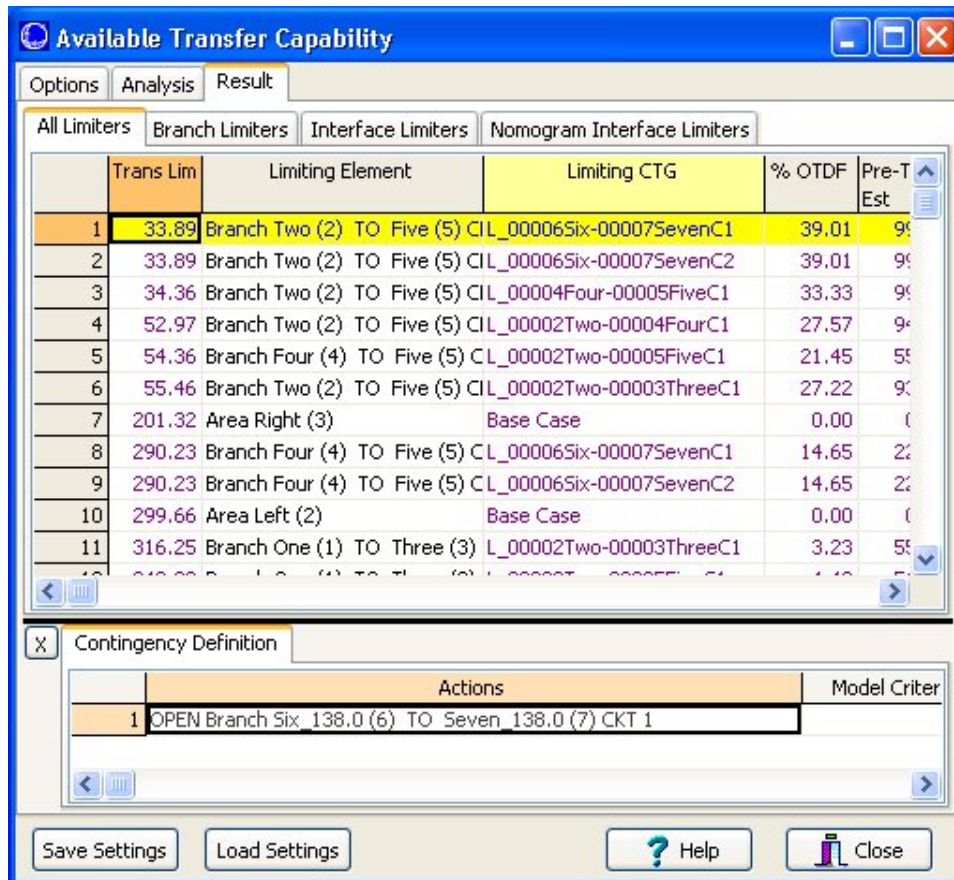
**Increase Transfer**

Click this button to open the Ramp Transfer Up dialog in order to increase (or decrease) the transfer level manually. This transfer remains in the system state if the dialog is closed without restoring the initial state.

Additional analysis can be done on a case that has had the transfer increased by using the ATC tool; however, the control method used in the ATC tool may differ from that in use once the ATC tool has been exited. When injection groups are used to increase the transfer, the AGC method that is used is island-based AGC. If this same method is not used for subsequent analysis, the dispatch and system state will be modified from the state with the transfer applied. When using areas or super areas for increasing the transfer, the area or super area is put on participation factor control for the ramping. The same problem can exist as with the injection groups if the same AGC method is not in use with subsequent analysis after the transfer ramping.

## ATC Dialog: Result Tab

When performing ATC Analysis on a single system state, the Result Tab is visible. The Result Tab consists of two sections: A [Transfer Limiters Display](#) and a [Contingency Definition Display](#).



### Result Tab

The Transfer Limiters Display contains tabbed sheets containing information on Branch, [Interface](#), and [Nomogram](#) Limiters. The user can also choose to display All Limiters. See [Transfer Limiters](#) for more information.

The Contingency Definition section displays information on the limiting contingency for selected transfer limiter if the user checked **Include Contingencies** on the [Common Options](#) tab.

## ATC Extra Monitors Dialog

Simulator's ATC tool determines the maximum amount of MW transfer between the seller and buyer. If you would like to also determine the flow on additional lines or interfaces at the transfer levels determined by the ATC tool, you can utilize ATC Extra Monitors. To open the Extra Monitors dialog, click the **Define Extra Monitors** button on the [Advanced Options](#) tab of the ATC Analysis Dialog.

The ATC Extra Monitors Display lists all of the ATC Extra Monitors defined. This list display is a [Case Information Display](#), and therefore has the same functionality as other common displays. See **Adding and Removing Extra Monitors** below.

	ATC ExMon Desc	ATC ExMon Obj	Monitor Limit	Relative Monitor Sensitivity Constant
1	MW flow	Branch Three (3) TO Four (4) CKT 1	100.00	none

### Extra Monitors Dialog

The default fields shown on this display are:

#### ATC ExMon Desc

This is a description of the monitored value. Presently, this is always MW flow.

#### ATC ExMon Obj

The power system element that is being monitored. This will be either a transmission branch or an interface.

#### Monitor Limit

This is the MW limit of the element being monitored.

#### Relative Monitor

Set this to a positive value to further filter the Transfer Limitations reported on the [Transfer Limiters Display](#). By default, Relative Monitor is set to *none*, and no additional filtering of limitations is performed. If this value is greater than zero, only Transfer Limitations that meet the following condition are included in the results.

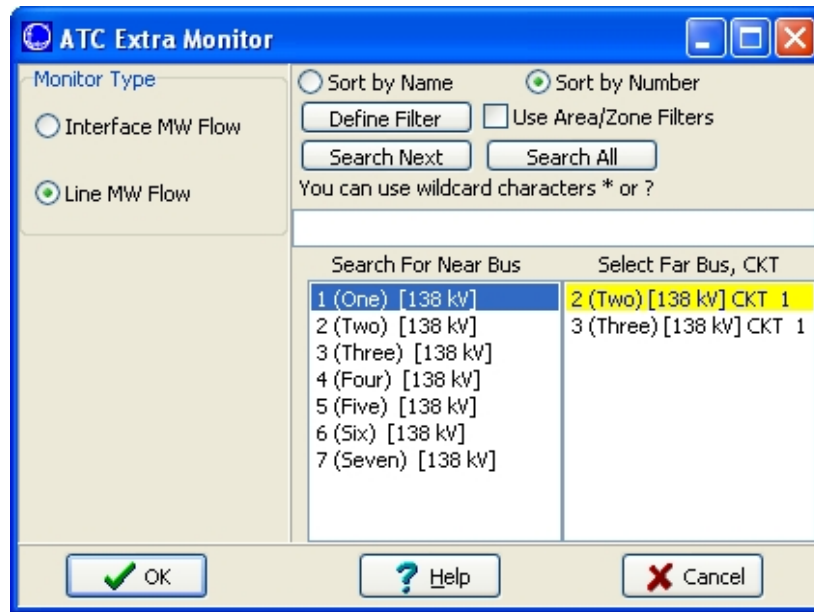
$$\frac{OTDF_{ELEMENT}}{PTDF_{ELEMENT}} \times \frac{ExtraMonRating}{Element Rating} \geq RelativeMonitor$$

This provides a measure of how much an interface or branch is affected by the transfer relative to its MW limit.

### Adding and Removing Extra Monitors

To delete an Extra Monitor, right-click on the desired record on the list display and select **Delete**.

To insert an Extra Monitor, right-click on the list display and select **Insert**. This opens the insert dialog.



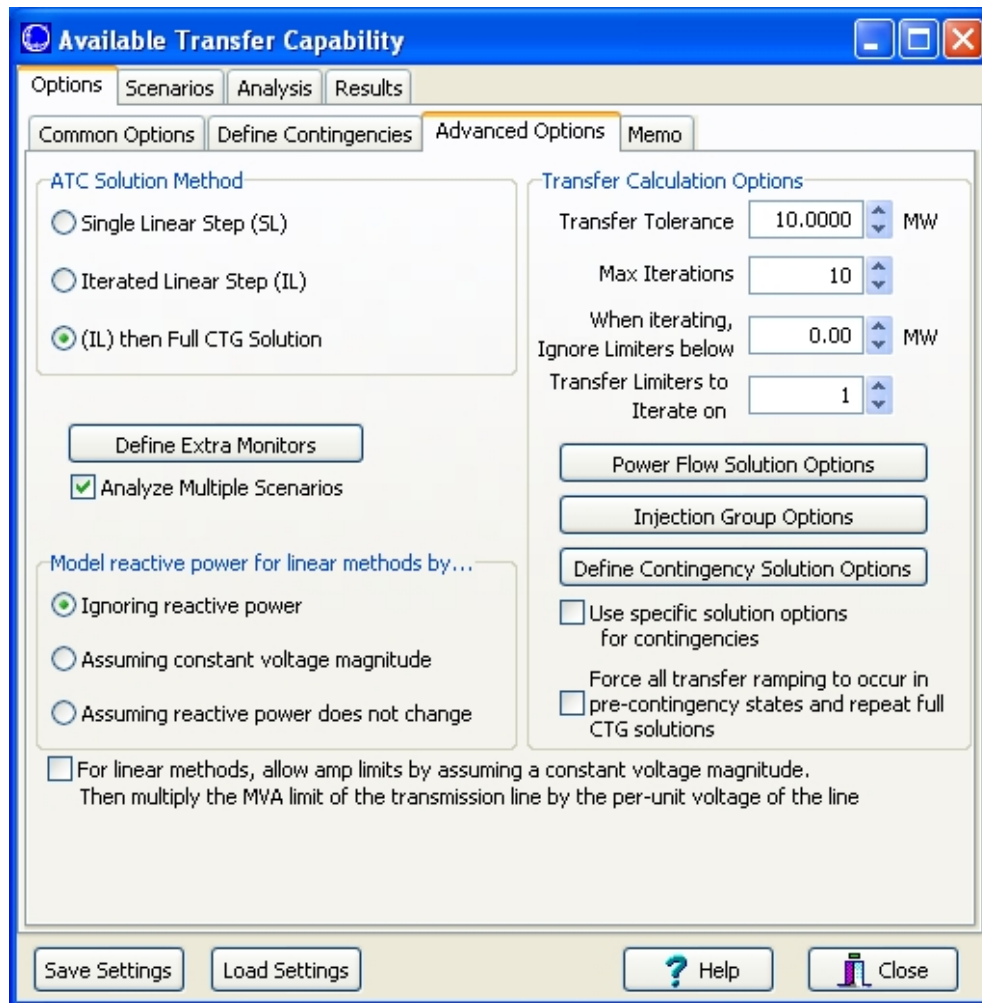
**Insert Extra Monitors Dialog**

Choose whether you want to monitor an Interface or Line MW flow. Next choose the interface or branch to be monitored. Note that the insert dialog allows the use of Simulator's [Advanced Search Engine](#) and techniques to aid in locating the desired interface or branch. Click **OK** to insert the record.

## Multiple Scenario Available Transfer Capability Dialog

To perform [Available Transfer Capability Analysis](#) for several system scenarios, check **Analyze Multiple Scenarios** on the **Advanced Options** tab of the [Available Transfer Capability Dialog](#). When Analyze Multiple Scenarios is not checked, the available tabs are [Options](#), [Analysis](#) and [Result](#). When Analyze Multiple Scenarios is checked then the following changes occur:

- [Scenarios Tab](#) appears
- [Results Tab](#) appears
- Result Tab is removed (replaced by Results)



**Analyze Multiple Scenarios Dialog**

By defining multiple scenarios, Simulator allows you to calculate ATC values for several different power system states automatically. Scenarios can be modified along three axes:

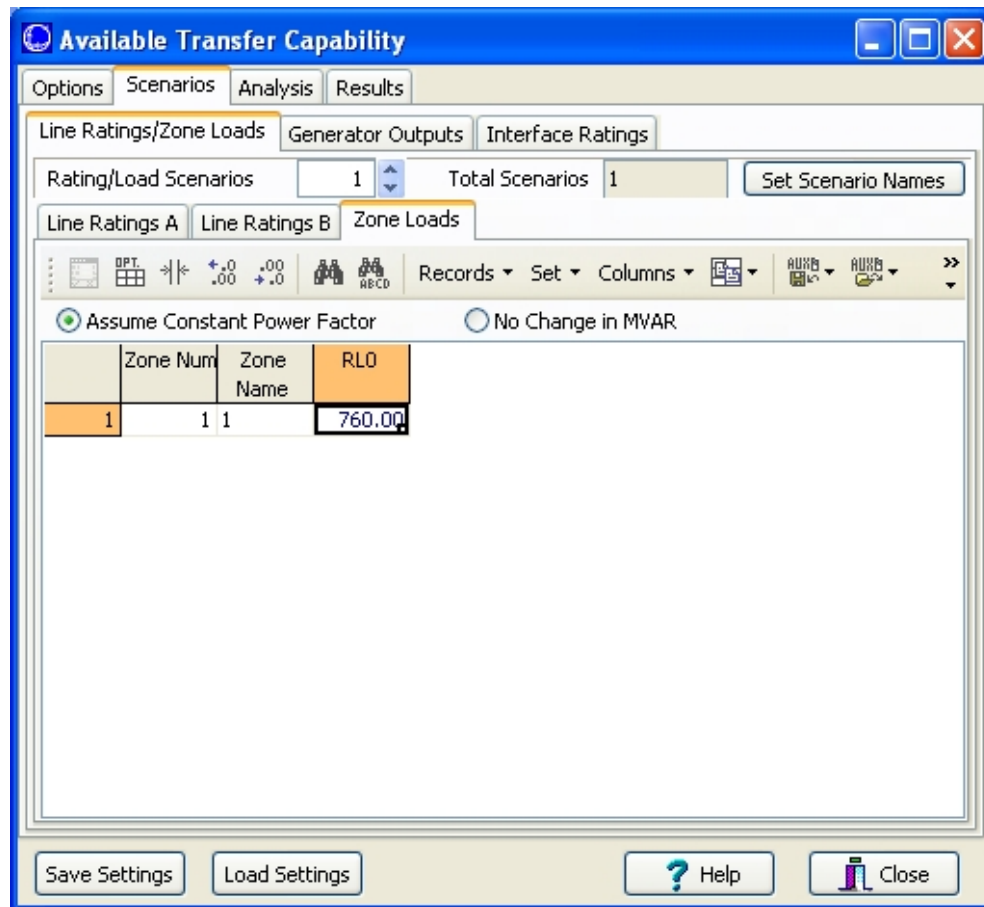
- Line Rating/Zone Load Scenarios (weather-related scenarios)
- Generation Scenarios (generation profiles)
- Interface constraints

See [Scenarios Tab](#) for more information.

## Multiple Scenario ATC Dialog: Scenarios Tab

By defining multiple scenarios, Simulator allows you to calculate ATC values for several different power system states automatically. Scenarios can be modified along three axes:

- Line Rating/Zone Load Scenarios (weather-related scenarios)
- Generation Scenarios (generation profiles)
- Interface constraints



Analyze Multiple Scenarios Dialog: Scenarios Tab

Each tab contains a list of the power system elements that will be modified during different scenarios. These lists are a familiar [Case Information Display](#) providing the same functionality as other displays

To insert a new power system element into the list, right-click on the list (below the headings) and choose **Insert**.

The Scenarios Tab has the following controls:

### Tabs and Sub-Tabs

The Scenarios Tab contains three tabbed sheets: **Line Ratings/Zone Loads**, **Generator Outputs** and **Interface Ratings**.



Line Ratings and Zone Loads are varied together when analyzing scenarios. This was chosen because they often vary together as a function of the weather. Notice that three sub-tabs (labeled Line Ratings A, Line Ratings B, and Zone Loads) appear at the bottom of the Line Ratings/Zone Loads tab. The A rating is typically used as a base case ratings, whereas the B limit is typically used as a line rating under contingency. The way line ratings are utilized is defined in [Limit Monitoring Settings](#).

### Number of Defined Scenarios per Element Type

On each tab, you may enter how many different scenarios should be defined for that kind of power system element. For instance if you set Generation Scenarios to 5, then list display on the Generation Tab will provide 5 columns labeled G0, G1, G2, G3, and G4. Generation outputs should then be entered into each cell representing the generation output in each scenario.

### Total Scenarios

Once you have specified the scenarios, Simulator is able to perform ATC Analysis on every combination of the axes. For example, assume you have the following:

- 10 sets of line ratings and zones load
- 8 sets of generation profiles
- 3 interface constraints

This yields a total of 240 different scenarios to calculate ( $10 \times 8 \times 3 = 240$ ). Be warned that that the more scenarios you analyze the longer the computation will take.

### Set Scenario Names

This button will bring up the Scenario Names dialog, where the user is able to assign a different name to the scenarios.

### Assume Constant Power Factor

This option is only available in the **Zone Loads** sub-tab of the **Line Ratings/Zone Loads** tab. If selected, it specifies that the Mvar will be ramped as the load MW is ramped, maintaining a constant power factor.

### No Change in MVAR

This option is only available in the **Zone Loads** sub-tab of the **Line Ratings/Zone Loads** tab. If selected, it specifies that the load Mvar is kept constant, independently of the variations of the load MW.

### Modified Elements

Each tab contains a list of the power system elements that will be modified during different scenarios. These lists are a familiar [Case Information Display](#) providing the same functionality as other displays.

To insert a new power system element into the list, right click on the list (below the headings) and choose **Insert**.

If a power system element does not show up in one of the lists of elements for a particular scenario, then it will retain its base case value during that scenario.

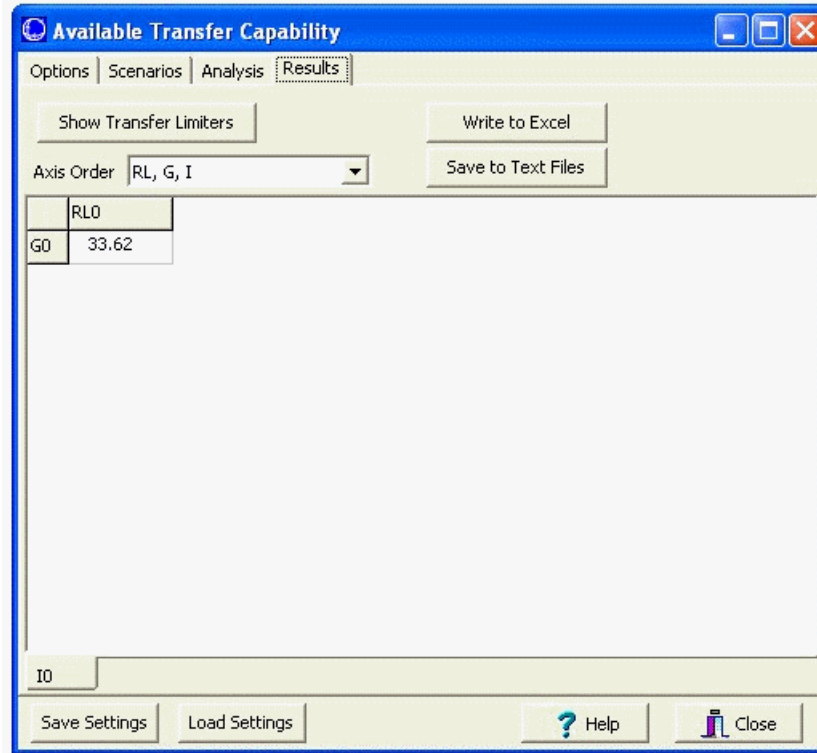
When saving or loading scenario elements in an auxiliary file, the elements are usually identified by primary key fields. Optionally, these elements can be identified by label. See the [Label](#) topic for more information about this.

---

## Multiple Scenario ATC Dialog: Results Tab

---

Once you have defined your scenarios and started the ATC Analysis, you can switch to the Results Tab to see the progress that ATC Analysis is making.



### Multiple Scenario Results Tab

The primary part of the Results tab contains a spreadsheet-like look-up table display. The layout of the display is dictated by the Axis Order selected. Note that because there are three axes possible, the row and column entries represent the first two axes while the third axis is represented by tabs at the bottom of the table. See [Local Menu Options](#) for information regarding the options available when right-clicking within the table.

This dialog has the following controls:

#### Show Transfer Limiters Button

Click the **Show Transfer Limiters** button to view the [Transfer Limiters](#) found under each scenario. This will open a separate dialog that displays a list of the Transfer Limiters. To see the Transfer Limiters for a particular scenario, click on the workbook cell that represents the scenario you are interested in and the separate dialog will update appropriately.

#### Axis Order

This menu is used to select the desired axis order. The three axes correspond to the three Scenarios Tabs.

- One axis has heading labels G0, G1, ... for the Generation Scenarios,
- Another with heading labels RL0, RL1, ... for the Rating/Load Scenarios,
- A third with heading labels I0, I1, ... for the Interface Scenarios.

**Results Display**

The primary part of the Results tab contains a spreadsheet-like look-up table display. The values shown in the table for each scenario are the most limiting Transfer Limit in MW.

**Write to Excel**

This button will send the results to an Excel spreadsheet.

**Save to Text Files**

This option will allow the user to save the results in an auxiliary file.

---

### **Multiple Scenario ATC Analysis - Results Tab: Local Menu Options**

---

If you right-click on the table contained on the Results Tab, you will bring up a local menu containing several options. The first several options are only enabled if you right-click on a cell representing a scenario. These options will perform an operation with respect to the Scenario:

**Take me to Scenario ...**

Implements the changes in the Scenario resulting in a modified system state.

**Determine Transfer Limit For Scenario ...**

Calculates the ATC for the Scenario, and then sets the system state back to the Initial State.

**Take me to the Transfer Limit For Scenario ...**

Performs ATC analysis for the Scenario, and then ramps the transfer to this limit.

Other options on this local menu are not related to the Scenario you have clicked on.

**Increase Transfer for Present System State**

Increments the transfer level for the present system state by a user-defined amount.

**Return to Initial State**

Returns the system state to the Initial State.

## Transfer Limiters Display

The Transfer Limiters Display shows the results of [ATC Analysis](#). This display appears either on the [Result Tab](#) of the [Available Transfer Capability Dialog](#) or as a separate window when performing [Multiple Scenario Available Transfer Capability Analysis](#).

The display contains four different tabs: All Limiters, Branch Limiters, Interface Limiters, and Nomogram Interface Limiters.

The Contingency Definition Display at the bottom of the window shows information on the defined contingency that caused the limitation selected from one of the four tabbed pages of limiters. For more information on this display, see [Contingency Analysis - Contingency Definition Display](#).

**Available Transfer Capability**

Options Analysis Result

Records Set Columns Options

All Limiters Branch Limiters Interface Limiters Nomogram Interface Limiters

	Trans Lim	Limiting Element	Limiting CTG	% OTDF	Pre-Trans Est	Limit Used
1	34.46	Branch Two (2) TO Five (5) CIL_00006Six-00007SevenC1		39.01	86.56	100.00
2	34.46	Branch Two (2) TO Five (5) CIL_00006Six-00007SevenC2		39.01	86.56	100.00
3	35.05	Branch Two (2) TO Five (5) CIL_00004Four-00005FiveC1		33.33	88.32	100.00
4	53.19	Branch Two (2) TO Five (5) CIL_00002Two-00004FourC1		27.57	85.34	100.00
5	55.31	Branch Four (4) TO Five (5) CIL_00002Two-00005FiveC1		21.45	48.14	60.00
6	55.77	Branch Two (2) TO Five (5) CIL_00002Two-00003ThreeC1		27.22	84.82	100.00
7	291.46	Branch Four (4) TO Five (5) CIL_00006Six-00007SevenC1		14.65	17.30	60.00
8	291.46	Branch Four (4) TO Five (5) CIL_00006Six-00007SevenC2		14.65	17.30	60.00
9	307.51	Branch One (1) TO Three (3) L_00002Two-00003ThreeC1		3.23	55.05	65.00
10	337.03	Branch One (1) TO Three (3) L_00002Two-00005FiveC1		4.49	49.87	65.00
11	353.02	Branch Four (4) TO Five (5) CIL_00001One-00002TwoC1		9.59	26.13	60.00

Contingency Definition

	Actions	Model Criteria	Status	Comments
1	OPEN Branch Six_138.0 (6) TO Seven_138.0 (7) CKT 1		CHECK	

Save Settings Load Settings Help Close

## Transfer Limiters Display

The Transfer Limiters Display has the following controls:

### All Limiters Tab

The All Limiters tab shows a list of all the transfer limitations found. This includes limitations on branches, interfaces, areas, zones, etc...

### Branch Limiters Tab

The Branch Limiters tab only shows those limitations with a transmission line or transformer as the limiting element. The limiting element description for the Branch Limiters tab is replaced by the bus numbers and names of the limiting element.

## Interface Limiters Tab

The Interface Limiters Tab only shows those limitations with an interface as the limiting element. The limiting element description is replaced by the name of the interface.

## Nomogram Interface Limiters Tab

The Nomogram Interface Limiters Tab only shows those limitations with a nomogram interface as the limiting element. The limiting element description is replaced by the name and segment of the nomogram interface.

## Limiters Fields

### Transfer Limit Field

The Transfer Limit Field shows the Transfer Limit in MW for the Limiting Element during the Limiting Contingency. This value depends on the ATC Solution Method Used:

- **Single Linear Step (SL):** Only one Linear ATC step is performed. The Transfer Limit values are those found during this step.
- **Iterated Linear Step (IL):** The Single Linear Step method is iterated during this method. The Transfer Limiters shown are those found during the final step performed. The actual Transfer Limit values are the values found at the last step plus the accumulated amount the transfer has been ramped. The Transfer Limitation(s) that were iterated on individually are highlighted in yellow.
- **Iterated Linear Step (IL) then Full CTG:** The Transfer Limits shown for the Transfer Limitation(s) that were used when iterating are equal to the accumulated amount that the transfer was ramped during the initial Iterated Linear Step and then the contingency analysis step. These limitations are highlighted in cyan. The Transfer Limits shown for all other limiters are the results of the initial Iterated Linear Step calculation performed before iterating on individual limitations for the contingency analysis.

For the iterated methods there are color indicators used to highlight the Transfer Limitations when certain situations are encountered:

- Fuchsia – Indicates that the power flow failed to converge as the transfer is ramped out in the iterated methods. The Transfer Limit that is reported is the accumulated transfer amount that has been ramped as of the last successful solution. When this situation is encountered, the top five dV/dQ sensitivities of all monitored buses in the case are also reported in the Transfer Limiters list. Limiting Element field will contain POWERFLOW DIVERGENCE.
- Lime Green – Indicates that the number of iterations has exceeded the maximum number of allowed iterations. The Transfer Limit that is reported is the accumulated transfer amount that has been ramped as of the last successful solution. Limiting Element field will contain OSCILLATING.
- Yellow – Indicates the Transfer Limitation(s) that were iterated on individually during the Iterated Linear Step (IL) method. Iteratively Found field will contain YES.
- Cyan – Indicates the Transfer Limitation(s) that were iterated on individually during the Iterated Linear Step (IL) then Full CTG method. Iteratively Found field will contain FULL.
- Orange – Indicates that the contingency failed to solve at some point during the Iterated Linear Step (IL) then Full CTG method. The Transfer Limit value that is reported is the accumulated transfer amount that has been ramped as of the last successful solution. Iteratively Found field will contain CTG\_FAIL\_IN\_FULL.
- Gray – Indicates that the power flow failed to converge while ramping the transfer amount during the Iterated Linear Step (IL) then Full CTG method while forcing the transfer ramping to occur pre-contingency. This indicates that the solution failed because of the transfer amount being ramped and not because the contingency

failed to solve. The Transfer Limit that is reported is the accumulated transfer amount that has been ramped as of the last successful solution. When this situation is encountered, the top five dV/dQ sensitivities of all monitored buses in the case are also reported in the Transfer Limiters list. Iteratively Found field will contain RAMP\_FAIL\_IN\_FULL.

#### **Limiting Element Field**

Shows a text description of the limiting element.

#### **Limiting CTG Field**

Shows the name of the limiting contingency.

#### **% OTDF Field**

This is the OTDF (or PTDF if the Limiting CTG is *Base Case*) on the Limiting Element for the transfer direction that is being studied. In other words, this is a linear estimate of the percent of the transfer that will appear on the Limiting Element if the Limiting CTG occurs.

**Note:** For iterated techniques, this is the PTDF or OTDF at the last Linear Iteration.

#### **Pre-Transfer Value Estimate Field**

If a contingency is not included in the Limiter, this is equal to the Initial Value. When a contingency is included in the Limiter, this is the linear estimate of the post-contingency flow before any transfer occurs. See [Available Transfer Capability Analysis](#).

**Note:** For iterated techniques, this is the estimate at the last Linear Iteration.

#### **Limit Used Field**

This is the value of the Limit being used by the ATC for the Limiting Element during the Limiting CTG. It reflects what is specified in [Limit Monitoring Settings](#) and options set on the [Advanced Options](#) tab.

#### **Contingency Definition Display**

This display shows information on the defined contingency that caused the limitation selected from one of the four tabbed pages of limiters. For more information on this display, see [Contingency Analysis - Contingency Definition Display](#).

---

## **ATC Analysis Methods - Solution Methods**

---

Simulator provides three methods of determining ATC.

- [Single Linear Step \(SL\)](#)
- [Iterated Linear Step \(IL\)](#)
- [Iterated Linear Step \(IL\) then Full Contingency Solution](#)



---

**ATC Analysis Methods - Single Linear Step (SL)**

---

The Single Linear Step approach is the most common ATC method. This method of ATC analysis uses only information about the present system state and sensitivities (mathematical derivatives) about the present system state. These sensitivities are embodied in the [PTDF](#) and [LODF](#) calculations.

Consider a transmission line with a limit of 10 MW, present loading of 5 MW and a PTDF of 10%. The estimated maximum transfer without causing an overload on the line is

$$\text{Transfer Limitation} = (\text{Limit} - \text{Present Loading}) / \text{PTDF} = (10 - 5) / 0.1 = 50 \text{ MW}$$

When including contingency analysis, the [OTDF \(Outage Transfer Distribution Factor\)](#) and linearized estimates of post-contingency flows are used to determine the Transfer Limitation.

$$\text{Transfer Limitation} = (\text{Limit} - \text{Post-Contingency Loading}) / \text{OTDF}$$

If we find the Transfer Limitation for every transmission branch (and interface) during each contingency, then the ATC is equal to the smallest Transfer Limit value.

**Note:** Simulator also monitors the possibility that a transfer will reduce the flow on a line until the line reaches its limit for flow in the opposite direction.

---

## ATC Analysis Methods - Iterated Linear Step (IL)

---

The Single Linear Step is an extremely fast method for determining the ATC. However, because it only uses present operating point information, controller changes are not taken into account. The linearization assumes that all controllers are fixed. The Iterated Linear Step (IL) method provides an alternative to the Single Linear Step that allows controller changes, but still performs its analysis in a reasonable amount of time. The (IL) method operates as follows:

- Perform Single Linear Step
- Set the Stepsize = Minimum Transfer Limitation found in Single Linear Step which is greater than the specified **When iterating, Ignore Limiters below** value
- If [abs(Stepsize) <= Tolerance] or [Iterations >= Max Iterations] then stop
  - o If the maximum number of iterations is reached, a transfer limitation will be reported highlighted in lime green indicating that the iterated process is oscillating. The Transfer Limit that is reported is the accumulated transfer amount that has been ramped as of the last successful solution.
  - o The iteration count is reset at the beginning of the iteration process.
- Implement transfer by amount of Stepsize and resolve power flow
  - o The power flow is solved according to the options set with the [Power Flow Solution Options](#).
  - o When calculating a transfer between injection groups, the [Island-Based AGC Power Flow Solution Options](#) applicable to dispatch using an injection group are used when determining which generators and loads will participate in the transfer. These options include whether or not AGC status should be considered when determining which units will participate in the ramping, whether or not to enforce generator MW limits, and whether or not to allow negative loads. The power factor to use when adjusting load is also specified with these options.
  - o To prevent oscillation of the solution, the Stepsize may be limited if the sign of the Stepsize changes throughout the iterations. At each iteration, the direction of the Stepsize is checked and the Stepsize is bounded based on the accumulated transfer at each iteration. The accumulated transfer is the total amount that the transfer between the seller and buyer has been ramped. The accumulated transfer is updated with the current Stepsize once it has been determined that Stepsize can be achieved. At the start of the iterated process, LowBound is initialized to a large negative number and HighBound is initialized to a large positive number. At each iteration the following checks are performed:
    - If Stepsize > 0 then LowBound = (Accumulated Transfer)
    - Else If Stepsize < 0 then HighBound = (Accumulated Transfer)
    - If Stepsize >= (HighBound - Accumulated Transfer)\*0.7
    - Then Stepsize = (HighBound - Accumulated Transfer)\*0.7
    - Else If Stepsize <= (LowBound - Accumulated Transfer)\*0.7
    - Then Stepsize = (LowBound - Accumulated Transfer)\*0.7
  - o During the implementation of the transfer, if the full Stepsize cannot be achieved, an attempt is made to implement as much of the transfer as possible in smaller stepsizes. The first attempt is to cut the Stepsize in half. If that succeeds then an iterated process occurs where the difference between the original full Stepsize and the successful stepsize is ramped. This process continues with the incremental stepsizes until as much of the original Stepsize ramping as possible is achieved.
  - o The ramping of the transfer is done so that each step adjusts injection from a common starting point so that the ramping is effectively done in the same direction even if the Stepsize backs off the transfer. At the start of the iterated process, the

base case generation and load values are stored. At each step where transfer ramping occurs, the generation and load values are returned to their base case levels and the transfer ramping is implemented from that point. As an example, take the situation where 1000 MW is ramped. The next Stepsize is calculated to be -100 MW. The total transfer amount achieved is then 900 MW. If ramping does not proceed in the same direction, generators that hit maximum limits when ramping out to 1000 MW will back off from these limits when ramping the -100 MW Stepsize. Ramping out to the total transfer amount of 900 MW in one step causes generators to hit their limits and stay there. If the ramping does not proceed in the same direction, ramping out by the total amount in one step and ramping by backing off the transfer will yield different results because some generators will no longer be at limits when ramping in steps. Ramping in the same direction is intended to prevent this discrepancy with generators at limits.

- o If the power flow fails to converge at this step, the iterated process will stop and a transfer limitation will be reported highlighted in fuchsia. The Transfer Limit that is reported is the accumulated transfer amount that has been ramped as of the last successful solution.
- o Reserve limits on generators and loads participating in the transfer will be checked if the option to enforce limits is active for the selected transactors. Reverse limits are checked regardless of how the option to **Report Generation Reserve Limits** is set with the [Common Options](#). For areas and super areas this means that the global enforcement of generator limits is active, the area enforcement of limits is active, and there is at least one generator whose enforcement of limits is active and this generator is on AGC control. For injection groups this means that the enforcement of limits for the injection group is active. When a reserve limit is found, the current iterations will stop and a transfer limit will be reported with the reserve limit and the area, super area, or injection group that is at a limit. If a reserve limit is encountered during the step where all transfer limiters are iterated on together, the entire ATC process will stop. If a reserve limit is encountered during the step where a limiter is being iterated on individually, the iterations will only stop for that limiter; other limiters will be processed if they do not hit a reserve limit as well.
- At new operating point, go back to first step and repeat

As the initial iteration process is performed, the transfer limiter may be different (i.e. the contingency and limiting element pair may be different) during each iteration. Once the Stepsize is determined to be less than or equal to the tolerance (or the maximum number of iterations is met) the order of the limiters is set.

At this point, the first user specified **Transfer Limiters to Iterate On** are taken from the list of limiters and the above process is repeated for each limiter individually, with the contingency and limiting element pair of each limiter remaining set as determined in the initial process. Transfer limitations for transfer limits found in this manner will be highlighted in yellow in the results.

This method takes into account controller changes that occur as you ramp out to the transfer level, but still avoids the full simulation of contingencies.

This method can be applied between combinations of areas and super areas, OR between two injection groups. Combinations of areas/superareas and injection groups are not allowed.

---

## ATC Analysis Methods - Iterated Linear Step (IL) then Full CTG Solution

---

This method takes the solution process a step further than just the Iterated Linear (IL) method. Instead of using linear sensitivities to calculate the impact of contingencies on the transfer limits, full contingencies are implemented and solved with a full power flow solution. Be aware however, that this calculation method can be extremely slow.

The initial iteration process of (IL) then Full CTG Solution method starts by performing the steps outlined in the [Iterated Linear \(IL\)](#) process. As the initial iteration process is performed, the transfer limiter may be different (i.e. the contingency and limiting element pair may be different) during each iteration. Once the Stepsize is determined to be less than or equal to the tolerance (or the maximum number of iterations is met) the order of the limiters is set.

At this point, the first user specified **Transfer Limiters to Iterate On** are taken from the list of limiters and the steps outlined in the [Iterated Linear \(IL\)](#) process are repeated for each limiter individually, with the contingency and limiting element pair of each limiter remaining set as determined in the initial process.

Once the iterated process has been repeated for an individual limiter, there are two different ways that the processing of the full contingency can proceed. If choosing to **NOT Force all transfer ramping to occur in pre-contingency states and repeat full CTG solutions**, the contingency of each transfer limiter is then enforced in the power flow topology, and a power flow solution is performed to give a new contingency power flow solution to use as the base for each transfer limiter. If the option to **Use Specific Solution Options for Contingencies** is checked, then the power flow is solved with the options defined with the [Contingency Solution Options](#). Otherwise, the power flow is solved with the options specified with the [Power Flow Solution Options](#). In either case, the [Make-Up Power](#) option specified with the contingency analysis options is used when applying the contingency. The iterative process outlined in the [Iterated Linear \(IL\)](#) method is then repeated again for each individual transfer limiter with the specified contingency of each limiter actually in place in the power flow solution. If the contingency solution fails to converge, the transfer limitation will be reported highlighted in orange and the Transfer Limit that is reported will be the value determined from the iterated process performed on the individual limiter before the contingency was applied.

If choosing to **Force all transfer ramping to occur in pre-contingency states and repeat full CTG solutions**, after the iterated process is complete for an individual limiter, the contingency of each transfer limiter is fully implemented in the following steps:

- Store the system state following the iterated process on the individual limiter
  - o At this point the accumulated transfer amount is the total amount that has been transferred following completion of the iterated process on the individual limiter.
  - o The Stepsize is zero for the first iteration.
- Apply the contingency and perform full power flow solution
  - o If the option to **Use Specific Solution Options for Contingencies** is checked, then the power flow is solved with the options defined with the [Contingency Solution Options](#). Otherwise, the power flow is solved with the options specified with the [Power Flow Solution Options](#). In either case, the [Make-Up Power](#) option specified with the contingency analysis options is used when applying the contingency.
  - o If the option to **Iterate on failed contingency** is NOT checked and the power flow fails to solve, the iterated process for the individual limiter will stop and a transfer limitation will be reported highlighted in orange. The Transfer Limit that is reported is the accumulated transfer amount that has been ramped as of the last successful solution.

- o If the option to **Iterate on failed contingency** IS checked and the power flow fails to solve, a new iteration loop is entered to determine the highest transfer level at which the contingency can be solved. This loop attempts to determine a more accurate transfer level at which the contingency fails to solve rather than simply reporting the transfer level at which the last power flow was successful regardless of the current Stepsize. To start the iterative process, the current Stepsize is reduced by half. The ramping is performed and the contingency is solved. The Stepsize will remain the same as long as the contingency can be solved. If the contingency fails to solve, the Stepsize will be reduced by half again. The ramping will continue until the contingency no longer solves, the accumulated transfer during the process meets the original Stepsize, or the Stepsize becomes smaller than the Stepsize tolerance. An additional check is done if the contingency does solve during this process; the flow on the monitored element is checked to determine if it exceeds its limit. If it is overloaded, the Stepsize will be reduced by half until the monitored element is no longer over its limit. While this check is being done on the monitored element limit, the ramping continues out to the specified amount and the contingency is solved. During this entire process, the Stepsize is limited to be between zero and the Stepsize that was ramped when the contingency failed to solve for the first time. When the iterative process on the failed contingency completes, the iterations are done for this particular limiter. The Transfer Limit that is reported is the accumulated amount that is possible without overloading the monitored element and in which the contingency solves. The limiter will be reported highlighted in orange.
- o If the power flow solution is successful, update the accumulated transfer with the Stepsize.
- Perform Single Linear Step on the individual limiter
  - o The Stepsize is the Transfer Limit found for the individual limiter. If the Stepsize is less than the specified **When iterating, Ignore Limiters below** value minus the accumulated transfer amount, it is set to zero.
- If  $[\text{abs}(\text{Stepsize}) \leq \text{Tolerance}]$  or  $[\text{Iterations} \geq \text{Max Iterations}]$  then stop
  - o If the maximum number of iterations is reached, a transfer limitation will be reported highlighted in lime green indicating that the iterated process is oscillating. The Transfer Limit that is reported is the accumulated transfer amount that has been ramped as of the last successful solution.
  - o The iteration count is reset at the beginning of the contingency iteration process for each individual limiter.
- Restore system to the state stored in the first step
  - o This step removes the contingency that was implemented so that the transfer ramping occurs prior to the contingency being applied.
- Implement transfer by amount of Stepsize and resolve power flow
  - o The power flow is solved according to the options set with the [Power Flow Solution Options](#).
  - o When calculating a transfer between injection groups, the [Island-Based AGC Power Flow Solution Options](#) applicable to dispatch using an injection group are used when determining which generators and loads will participate in the transfer. These options include whether or not AGC status should be considered when determining which units will participate in the ramping, whether or not to enforce generator MW limits, and whether or not to allow negative loads. The power factor to use when adjusting load is also specified with these options.
  - o To prevent oscillation of the solution, the Stepsize may be limited if the sign of the Stepsize changes throughout the iterations. At each iteration, the direction of the Stepsize is checked and the Stepsize is bounded based on the accumulated transfer at each iteration. The accumulated transfer is the total amount that the transfer

between the seller and buyer has been ramped. The accumulated transfer is updated with the current Stepsize once it has been determined that Stepsize can be achieved. At the start of the iterated process, LowBound is initialized to a large negative number and HighBound is initialized to a large positive number. At each iteration the following checks are performed:

- If Stepsize > 0 then LowBound = (Accumulated Transfer)
- Else If Stepsize < 0 then HighBound = (Accumulated Transfer)
- If Stepsize >= (HighBound - Accumulated Transfer)\*0.7
- Then Stepsize = (HighBound - Accumulated Transfer)\*0.7
- Else If Stepsize <= (LowBound - Accumulated Transfer)\*0.7
- Then Stepsize = (LowBound - Accumulated Transfer)\*0.7
- o During the implementation of the transfer, if the full Stepsize cannot be achieved, an attempt is made to implement as much of the transfer as possible in smaller stepsizes. The first attempt is to cut the Stepsize in half. If that succeeds than an iterated process occurs where the difference between the original full Stepsize and the successful stepsize is ramped. This process continues with the incremental stepsizes until as much of the original Stepsize ramping as possible is achieved.
- o The ramping of the transfer is done so that each step adjusts injection from a common starting point so that the ramping is effectively done in the same direction even if the Stepsize backs off the transfer. Additional details about this are provided in the steps of the [Iterated Linear \(IL\)](#) process.
- o If the power flow fails to solve at this point, the iterated process for the individual limiter will stop and a transfer limitation will be reported highlighted in gray. This Transfer Limit that is reported is the accumulated transfer amount that has been ramped as of the last successful solution.
- o Reserve limits on generators and loads participating in the transfer will be checked if the option to enforce limits is active for the selected transactors. Reverse limits are checked regardless of how the option to **Report Generation Reserve Limits** is set with the [Common Options](#). For areas and super areas this means that the global enforcement of generator limits is active, the area enforcement of limits is active, and there is at least one generator whose enforcement of limits is active and this generator is on AGC control. For injection groups this means that the enforcement of limits for the injection group is active. When a reserve limit is found, the current iterations will stop and a transfer limit will be reported with the reserve limit and the area, super area, or injection group that is at a limit.
- Go back to second step and repeat

Transfer limitations found by implementing the full contingency solution are highlighted in cyan in the results.

This method can be applied between combinations of areas and superareas, OR between two injection groups. Combinations of areas/superareas and injection groups are not allowed.

---

## Simulator Automation Server (SimAuto) Overview

---

The SimAuto tool is only available if you have purchased the SimAuto add-on to the base Simulator package. [Contact PowerWorld Corporation](#) for details about ordering the SimAuto version of Simulator.

PowerWorld Corporation also offers Optimal Power Flow ([OPF](#)), Available Transfer Capability [ATC](#), Simulation Automation Server ([SimAuto](#)), and Security Constrained Optimal Power Flow ([SCOPE](#)) add-ons. For more information see.

SimAuto provides PowerWorld customers the ability to access PowerWorld Simulator functionality within a program written externally by the user. The Simulator Automation Server acts as a COM object, which can be accessed from various programming languages that have COM compatibility. Examples of programming tools with COM compatibility are Borland® Delphi, Microsoft® Visual C++, Microsoft® Visual Basic, and Matlab® (among others). For more information on COM Objects and Automation Servers, see the help for Microsoft Windows.

The Automation Server of Simulator works very well in combination with Simulator [Script Commands](#) and [Auxiliary Files](#). It is beneficial to become familiar with these topics when considering using the Simulator Automation Server.

Note that previous users of SimAuto in Version 9 will need to update their function calls to SimAuto functions. PowerWorld Corporation found it imperative to change the function calls for SimAuto, in order to remedy irreconcilable problems when using SimAuto with some programming languages. The documentation provided should provide adequate help on the changes needed, but as always, if any questions arise, please [contact](#) PowerWorld Corporation for more information.

---

## Installing Simulator Automation Server

---

Note that previous users of SimAuto in Version 9 will need to update their function calls to SimAuto functions. PowerWorld Corporation found it imperative to change the function calls for SimAuto, in order to remedy irreconcilable problems when using SimAuto with some programming languages. The documentation provided should provide adequate help on the changes needed, but as always, if any questions arise, please [contact](#) PowerWorld Corporation for more information.

Installing the [Simulator Automation Server](#) requires no additional steps beyond installing PowerWorld Simulator as normal. When a version of PowerWorld Simulator containing the Simulator Automation Server is installed on your computer, the install program automatically adds the information needed by the Simulator Automation Server to the registry.

If for some reason the registration fails, be sure you have the SimAuto add-on for Simulator and you have either run the application on the computer or run the command **pwrworld / regserver** at the command line. If instead you would like SimAuto to un-register itself as a COM object, run the command **pwrworld /unregserver** at the command line.



---

## Including Simulator Automation Server Functions

---

Before you can access the [functions](#) defined by the [Simulator Automation Server](#) when writing the code for your external program, you must first include the library of functions defined for the Simulator Automation Server. This kind of library is referred to as a Type Library, which describes the available functions in a manner that can be interpreted by different programming languages. Importing a Type Library from another program is usually fairly simple, but the procedure does vary depending on the programming tool you are using. Please see the help for your programming tool of choice on how to import a Type Library or COM functions from another program.

### Examples

The following examples are just a few specific examples for certain programming media. The procedure may be different for other programming media not listed. In addition, a procedure given for a certain type of programming media may be one variation from several possible procedures for accomplishing the same task.

#### Borland Delphi

- Install the version of PowerWorld Simulator with the Simulator Automation Server included.
- In Delphi, choose **Import Type Library...** from the Project menu.
- In the list of libraries, search for and choose pwrworld Library.
- If pwrworld Library is not in the list, click **Add**. Find and choose the Pwrworld.exe file from the PowerWorld Simulator directory, and click **Open**.
- You should see the class name TSimulatorAuto in the list of Class names.
- Click **Install** to include the PowerWorld Simulator Type Library.

#### Microsoft Visual Basic for Applications

- No additional tasks are necessary
- Importing Type Library still works (See [Including Functions for version 9](#)).

#### Microsoft Visual C++

- Install the version of PowerWorld Simulator with the Simulation Automation Server included.
- Add **#import "...\\powerworld.exe"** in your external program code, using the full path to the PowerWorld Simulator executable program.
- Add **using namespace pwrworld** in your external program code.

#### Matlab v.6.5 r.13

- No additional tasks are necessary.

---

## Connecting to Simulator Automation Server

---

Once the Type Library or COM functions have been included in your programming environment, the [Simulator Automation Server](#) can be handled as any other object in your code. The method for assigning and connecting to the Simulator Automation Server can vary depending on the programming environment used, but the idea is basically the same. You define a variable in your program to point to the server object, which is called SimulatorAuto. If the Type Library was imported properly, you should have full access to the SimulatorAuto object and its defined [functions](#). Again, the procedure for creating the object and connecting to SimulatorAuto may vary for different programming languages. Check the help for your programming environment on connecting to COM or Automation servers.

### Examples

The following examples are just a few specific examples for certain programming media. The procedure may be different for other programming media not listed. In addition, a procedure given for a certain type of programming media may be one variation from several possible procedures for accomplishing the same task.

#### Borland Delphi 5

- Add **pwrworld\_TLB** to the **uses** section of your unit.
- Declare a variable globally or as part of another object: **A : ISimulatorAuto**
- Initialize the variable: **A := nil**
- To connect to the Simulator Automation Server, create the connection: **A := CoSimulatorAuto.create**
- Perform function calls to the Simulator Automation Server: **Output := A.SomeFunction(parameters)**
- To close the connection to the Simulator Automation Server, remove the reference by again setting: **A := nil**

#### Microsoft Visual Basic for Applications

##### Early Binding:

- To connect to the Simulator Automation Server, create the connection initializing the variable:

**Dim A as New pwrworld.SimulatorAuto**

##### Late Binding:

- Declare a variable globally or as part of another object or function: **Dim A As Object**
- To connect to the Simulator Automation Server, create the connection:

**Set A = CreateObject("pwrworld.SimulatorAuto")**

##### Both Early and Late Binding:

- Perform function calls to the Simulator Automation Server: **Output = A.SomeFunction parameters**
- To close the connection to the Simulator Automation Server, remove the reference: **Set A = Nothing**
- If Type Library was imported, connection can also be achieved as in version 9 (See [Connecting to Simulator Automation Server in version 9](#)).

#### Microsoft Visual C++

- Declare a variable globally or as part of another object or function: **ISimulatorAutoPtr \*A**
- Declare a variable globally or as part of another object or function: **CLSID clsid**

- Declare a variable globally or as part of another object or function: **HRESULT hr**
- Obtain the class identifier (clsid) with the following command:  
**hr = CLSIDFromProgID(L"pwrworld.SimulatorAuto", &clsid)**
- Initialize variable A: **A = new IsimulatorAutoPtr**
- To connect to the Simulator Automation Server, create the connection:  
**hr = A>CreateInstance(clsid, NULL, CLSCTX\_SERVER)**
- Perform function calls to the Simulator Automation Server: **Output = A.SomeFunction(parameters)**
- To close the connection to the Simulator Automation Server, release the reference:  
**hr = A>Release()**

**Matlab v.6.5 r.13**

- To connect to the Simulator Automation Server, create the connection:  
**A = actxserver('pwrworld.SimulatorAuto')**
- Perform function calls to the Simulator Automation Server: **Output = A.SomeFunction(parameters)**
- To close the connection to the Simulator Automation Server, delete the connection:  
**delete(A)**

---

## **Passing Data to the Simulator Automation Server**

---

### **Passing Data to the Simulator Automation Server**

All data to the Simulator Automation Server is passed by value rather than by reference (in pointer terminology, this corresponds to sending data instead of pointer to data; in Microsoft Visual Basic®, this corresponds to sending data ByVal instead of ByRef). This makes the marshalling of data between client software and the COM object much easier.

### **No Optional Parameters**

There are no optional parameters for any of the Simulator Automation functions. All functions must be called with every argument filled.

---

## Getting Data from the Simulator Automation Server

---

### The Output Structure

Every function called on the SimulatorAuto object returns the same value, Output, which has a well-defined structure.

Output is of type **VARIANT**, and is an array of **VARIANTS**. Output is zero-indexed.

The first element *always* contains any errors occurring during execution. For those functions returning more than one element in the Output array (e.g. ListOfDevices), explanation is provided below when discussing the specific method.

### Error Handling

As mentioned above, the first item in the Output VARIANT array, Output[0], contains any errors occurring during the function's execution. If no errors occurred during the function's execution, Output[0] will be set to an empty BSTR (string) represented in most languages by either "" or "".

### Error Format

If an error string is returned, it will be in the following format:

[method name]: [error\_explanation]

e.g. RunScriptCommand: Error occurred processing script command - check command syntax

---

## Simulator Automation Server Functions

---

The following list of functions is currently available once the SimulatorAuto object is set in your code. Check the help sections on these functions to see more detail on the particular function.

[ChangeParameters](#)

[ChangeParametersSingleElement](#)

[ChangeParametersMultipleElement](#)

[ChangeParametersMultipleElementFlatInput](#)

[CloseCase](#)

[GetParametersSingleElement](#)

[GetParametersMultipleElement](#)

[GetParametersMultipleElementFlatOutput](#)

[GetParameters](#) (for compatibility with version 9)

[ListOfDevices](#)

[ListOfDevicesAsVariantStrings](#)

[ListOfDevicesFlatOutput](#)

[OpenCase](#)

[ProcessAuxFile](#)

[RunScriptCommand](#)

[GetFieldList](#)

[SaveState](#)

[LoadState](#)

[SaveCase](#)

[SendToExcel](#)

[WriteAuxFile](#)

---

## ChangeParameters Function

---

The ChangeParameters function has been replaced by the [ChangeParametersSingleElement](#) function. ChangeParameters can still be called as before, but will now just automatically call ChangeParametersSingleElement, and pass on the parameters to that function.

### Function Prototype

**ChangeParameters(ObjectType, ParamList, Values)**

### Parameter Definitions

**ObjectType : String** The type of object you are changing parameters for.

**ParamList : Variant** A variant array storing strings (COM Type BSTR). This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables for the specific device, or the device cannot be identified.

**Values : Variant** A variant array storing variants. This array can store any type of information (integer, string, etc.) in each array position. A value should be passed for each field variable given in the ParamList. The Values array must contain the [key field](#) values for the specific device, or the device cannot be identified.

---

## ChangeParametersSingleElement Function

---

The ChangeParametersSingleElement function allows you to set a list of parameters for a single object in a case loaded into the [Simulator Automation Server](#). In addition to changing parameters for objects, this function can also be used to set options for some of the Simulator tools, such as [ATC](#) and [OPF](#). This function is identical in setup to the [GetParametersSingleElement](#) function, with the exception that the Values array must contain a value for each field variable given in the ParamList array.

### Function Prototype

**ChangeParametersSingleElement(ObjectType, ParamList, Values)**

### Parameter Definitions

**ObjectType : String** The type of object you are changing parameters for.

**ParamList : Variant** A variant array storing strings (COM Type BSTR). This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables for the specific device, or the device cannot be identified.

**Values : Variant** A variant array storing variants. This array can store any type of information (integer, string, etc.) in each array position. A value should be passed for each field variable given in the ParamList. The Values array must contain the [key field](#) values for the specific device, or the device cannot be identified.

### Output

ChangeParametersSingleElement only returns the first element in Output, the error string.

### Notes

The ParameterList and Values arrays must be the same size, as each parameter must have a corresponding value to be assigned.



---

**ChangeParametersSingleElement Sample Code**


---

**Borland® Delphi**

```

Var ParamList, ValueList : OLEVariant

// Set ParamList up to modify the maximum number of iterations
// and the system base for the power flow simulations
ParamList := VarArrayCreate([1,2], varOleStr);
ParamList[1] := 'MaxItr';
ParamList[2] := 'SBase';

// ValueList is setup with 41 and 410 for MaxItr and SBase,
// respectively
ValueList := VarArrayCreate([1,2], varOleStr);
ValueList[1] := 41;
ValueList[2] := 410;

// Make the ChangeParametersSingleElement call
Output = SimAuto.ChangeParametersSingleElement('Sim_Solution_Options', _
ParamList, ValueList)

```

**Microsoft® Visual Basic for Applications**

```

' Set ParamList up to modify the maximum number of iterations
' and the system base for the power flow simulations
Dim ParamList As Variant
ParamList = Array("MaxItr", "Sbase")

' ValueList is setup with 41 and 410 for MaxItr and SBase,
' respectively
Dim ValueList As Variant
ValueList = Array(45, 90)

' Make the ChangeParametersSingleElement call
Output = SimAuto.ChangeParametersSingleElement("Sim_Solution_Options", _
ParamList, ValueList)

```

**Matlab®**

```

% Set ParamList up to modify the maximum number of iterations
% and the system base for the power flow simulations
ParamList = {'MaxItr' 'Sbase'};

% values is setup with 41 and 410 for MaxItr and SBase,

```

```
% respectively
values = [41 410];

% Convert the values matrix to a set of cells for passing
% through the COM interface
ValueList = num2cell(values);

' Make the ChangeParametersSingleElement call
Output = SimAuto.ChangeParametersSingleElement('Sim_Solution_Options', _
ParamList, ValueList)
```

---

## ChangeParametersMultipleElement Function

---

The ChangeParametersMultipleElement function allows you to set parameters for multiple objects of the same type in a case loaded into the [Simulator Automation Server](#). This function is very similar to the [ChangeParametersSingleElement](#), but allows for modifying multiple elements with a single function call. The advantage of this function is that it is much faster to change multiple elements with a single function call than it is to repeatedly call ChangeParametersSingleElement multiple times.

### Function Prototype

**ChangeParametersMultipleElement(ObjectType, ParamList, Values)**

### Parameter Definitions

**ObjectType : String** The type of object you are changing parameters for.

**ParamList : Variant** A variant array storing strings (COM Type BSTR). This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables for the specific device, or the device cannot be identified.

**ValueList : Variant** A variant array storing arrays of variants. This is the difference between the multiple element and single element change parameter functions. This array stores a list of arrays of values matching the fields laid out in ParamList. You construct ValueList by creating an array of variants with the necessary parameters for each device, and then inserting each individual array of values into the ValueList array. SimAuto will pick out each array from ValueList, and calls ChangeParametersSingleElement internally for each array of values in ValueList.

### Output

ChangeParametersMultipleElement only returns the first element in Output, the error string.

### Notes

The ParameterList and each array of values stored in ValueList must be the same size, as each parameter must have a corresponding value to be assigned.

---

**ChangeParametersMultipleElement Sample Code**


---

**Borland® Delphi**

```
Var ParamList, ValueList : OLEVariant
```

```
// Set ParamList up to modify the MW output of generators
ParamList := VarArrayCreate([1,3], varOleStr);
ParamList[1] := 'BusNum';
ParamList[2] := 'GenID';
ParamList[3] := 'GenMW';

// ValueList is setup with MW values for generators at buses 1 and 2
ValueList := VarArrayCreate([1,2], varOleStr);
For k := 1 to 2 do begin
  IndValueList := VarArrayCreate([1,3], varOleStr);
  IndValueList[1] := k;
  IndValueList[2] := '1';
  IndValueList[3] := k*10;
  ValueList[k] := IndValueList;
End;

// Make the ChangeParametersMultipleElement call
Output = SimAuto.ChangeParametersMultipleElement('Sim_Solution_Options', _
  ParamList, ValueList)
```

**Microsoft® Visual Basic for Applications**

```
' Set ParamList up to modify the MW output of generators
Dim ParamList As Variant
ParamList = Array("BusNum", "GenID", "GenMW")

' ValueList is setup with MW values for generators at buses 1 and 2
Dim ValueList(2) As Variant
For k = 0 to 1
  Dim IndValueList As Variant
  IndValueList = Array(k+1,"1",(k+1)*10)
  ValueList(k) = IndValueList
Next

' Make the ChangeParametersMultipleElement call
Output = SimAuto.ChangeParametersMultipleElement("Sim_Solution_Options", _
  ParamList, ValueList)
```



---

## ChangeParametersMultipleElementFlatInput Function

---

The ChangeParametersMultipleElementFlatInput function allows you to set parameters for multiple objects of the same type in a case loaded into the [Simulator Automation Server](#). This function is very similar to the [ChangeParametersMultipleElement](#), but uses a single dimensioned array of values as input instead of a multi-dimensioned array of arrays. The advantage of this function is that it is much faster to change multiple elements with a single function call than it is to repeatedly call [ChangeParametersSingleElement](#) multiple times. An additional advantage over [ChangeParametersMultipleElement](#) is that you can still take advantage of the speed improvement, even if the programming language you are using does not support multi-dimensioned arrays.

### Function Prototype

**ChangeParametersMultipleElement(ObjectType, ParamList, NoOfObjects, ValueList)**

### Parameter Definitions

**ObjectType : String** The type of object you are changing parameters for.

**ParamList : Variant** A variant array storing strings (COM Type BSTR). This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables for the specific device, or the device cannot be identified.

**NoOfObjects** You must pass an integer number of devices that are passing values for. SimAuto will automatically check that the number of parameters for each device (counted from ParamList) and the number of objects integer correspond to the number of values in value list (counted from ValueList.)

**ValueList : Variant** A variant array storing a list of variants. Value list can be an array with many values, as it is a single dimensioned array of all values for all devices that are being changed. The structure of the ValueList array is such that all of the parameters for the first object are listed first, then all parameters for the second object, and so on. The parameters must be in the same order as given in ParamList. In other words, your array would look like:

ValueList = Array(Obj1Param1, Obj1Param2, Obj2Param1, Obj2Param2, Obj3Param1, ..., ObjNParam1, ObjNParam2)

### Output

ChangeParametersMultipleElementFlatInput only returns the first element in Output, the error string.

### Notes

If the number of parameters given in ParamList multiplied by the number of objects passed does not equal the total number of values in ValueList, SimAuto will abort the function call.

---

**ChangeParametersMultipleElementFlatInput Sample Code**


---

**Borland® Delphi**

```
Var ParamList, ValueList : OLEVariant
```

```
// Set ParamList up to modify the MW output of generators
```

```
ParamList := VarArrayCreate([1,3], varOleStr);
```

```
ParamList[1] := 'BusNum';
```

```
ParamList[2] := 'GenID';
```

```
ParamList[3] := 'GenMW';
```

```
// ValueList is setup with MW values for generators at buses 1 and 2
```

```
NumObjects := 2;
```

```
NumFields := NumObjects * 3;
```

```
ValueList := VarArrayCreate([1,NumFields], varOleStr);
```

```
For k := 0 to 1 do begin
```

```
ValueList[3*k+1] := k;
```

```
ValueList[3*k+2] := '1';
```

```
ValueList[3*k+3] := k*10;
```

```
End;
```

```
// Make the ChangeParametersMultipleElementFlatInput call
```

```
Output = SimAuto.ChangeParametersMultipleElementFlatInput('Sim_Solution_Options', _  
ParamList, NumObjects, ValueList)
```

**Microsoft® Visual Basic for Applications**

```
' Set ParamList up to modify the MW output of generators
```

```
Dim ParamList As Variant
```

```
ParamList = Array("BusNum", "GenID", "GenMW")
```

```
' ValueList is setup with MW values for generators at buses 1 and 2
```

```
NumObjects = 2
```

```
NumFields = NumObjects * 3
```

```
Dim ValueList(NumObjects) As Variant
```

```
For k = 0 to 1
```

```
ValueList(3*k+1) = k
```

```
ValueList[3*k+2] := '1';
```

```
ValueList[3*k+3] := k*10;
```

```
Next
```

```
' Make the ChangeParametersMultipleElementFlatInput call
```

```
Output = SimAuto.ChangeParametersMultipleElementFlatInput("Sim_Solution_Options", _  
    ParamList, NumObjects, ValueList)
```



---

**CloseCase Function**

---

The CloseCase function is used to close a load flow case loaded in the [Simulator Automation Server](#). This function should be called at some point after the [OpenCase](#) function.

**Function Prototype**

**CloseCase()**

**Parameter Definitions**

No parameters are passed.

**Output**

CloseCase returns only one element in Output—any errors which may have occurred when attempting to close the case.

---

**CloseCase Function: Sample Code**

---

**Borland® Delphi**

Output := SimAuto.CloseCase();

**Microsoft® Visual Basic for Applications**

Output = SimAuto.CloseCase()

**Matlab®**

Output = SimAuto.CloseCase

---

**GetFieldList Function**

---

The GetFieldList function is used to find all fields contained within a given object type.

**Function Prototype**

**GetFieldList(ObjectType)**

**Parameter Definitions**

**ObjectType : String** The type of object for which the fields are requested.

**Output**

GetFieldList returns two elements of the Output array. The first element, as with the other functions, returns any errors that might have occurred. The second element of the Output array contains an  $n \times 4$  array of fields. The layout of this array is virtually identical to the output obtained by going to **Help > Export Case Object Fields** from the main menu of Simulator. The first column, corresponding to the (n,0) column in the field array, specifies which fields are key fields for the object. The second column, (n,1), contains the internal name of the field. The third column, (n,2), contains the type of data stored in the string (e.g. String, Integer, Real). The fourth column, (n,3), contains the display-friendly name of the field.

---

**GetFieldList Function: Sample Code**

---

**Microsoft® Visual Basic for Applications**

```
Dim objecttype As String
```

```
' Object type to obtain  
objecttype = "branch"
```

```
' Make the GetField call  
Output = SimAuto.GetFieldList(objecttype)
```

**Matlab®**

```
% Object type to obtain  
objecttype = 'branch';
```

```
% Make the GetField call  
Output = SimAuto.GetFieldList(objecttype);
```

---

## GetParametersSingleElement Function

---

The GetParametersSingleElement function is used to request the values of specified fields for a particular object in the load flow case. For returning field values for multiple objects, you can use a loop to make repeated calls to the GetParametersSingleElement function, and pass the object and desired field information for each object. This function is identical in setup to the [ChangeParameters](#) function, with the exception that the Values array will be updated with the values for the field variables defined in ParamList.

### Function Prototype

**GetParametersSingleElement(ObjectType, ParamList, Values)**

### Parameter Definitions

**ObjectType : String** The type of object you are changing parameters for.

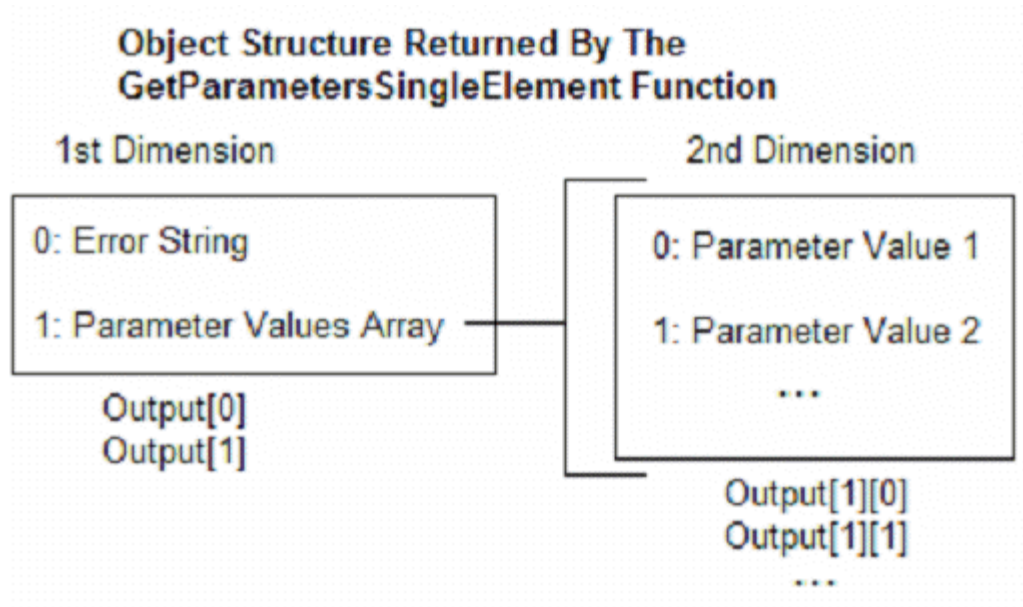
**ParamList : Variant** A variant array storing strings. This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables for the specific device, or the device cannot be identified. The remaining field variables in the array define which values to retrieve from Simulator.

**Values : Variant** A variant array storing variants. This array can store any type of information (integer, string, etc.) in each array position. Values must be passed for the [key field](#) variables in ParamList, in the same array position. The remaining field positions in the Values array should be set to zero.

### Output

GetParametersSingleElement returns both the first element in Output—containing any errors occurring during execution of the function—and a second element in Output. The second element returned in the Output structure is a one dimensional array containing the values corresponding to the fields specified in ParamList.

The Output structure of GetParametersSingleElement is shown in the following figure.



---

**GetParametersSingleElement Function: Sample Code Borland® Delphi**


---

```
// This example retrieves some parameters for bus 2 of the loaded
// case, using the GetParametersSingleElement function, as well as
// the old GetParameters function

procedure TMainForm.RunGPSEClick(Sender: TObject);
var
  Output : OLEVariant;
  FieldBusArray, ValueBusArray : OLEVariant;
  i : Integer;
begin
  // Declares fields array to be sent to Excel
  FieldBusArray := VarArrayCreate([1,5], varOleStr);
  FieldBusArray[1] := 'pwBusNum';
  FieldBusArray[2] := 'pwBusname';
  FieldBusArray[3] := 'pwBusKVVolt';
  FieldBusArray[4] := 'pwBusPUVolt';
  FieldBusArray[5] := 'pwBusAngle';
  ValueBusArray := varArrayCreate([1,5],varOleStr);
  ValueBusArray[1] := 2; // To get parameters for bus 2
  ValueBusArray[2] := 0;
  ValueBusArray[3] := 0;
  ValueBusArray[4] := 0;
  ValueBusArray[5] := 0;

  // Gets parameters with GetParametersSingleElement function
  Output := SimAuto.GetParametersSingleElement('bus', FieldBusArray, ValueBusArray);
  if (string(Output[0]) <> '') then
    StatusBar1.Panels[1].Text := 'Error: ' + string(Output[0])
  else
    begin
      StatusBar1.Panels[1].Text := 'Parameters got.';
      Memo1.Lines.Add('== GetParametersSingleElement ==');
      Memo1.Lines.Add('Value : Output[1][i]');
      for i := VarArrayLowBound(Output[1],1) to VarArrayHighBound(Output[1],1) do begin
        Memo1.Lines.Add(FieldBusArray[i] + ' : ' + string(Output[1][i]));
      end;
      Memo1.Lines.Add('');
    end;

  // Gets parameters with old function GetParameters
  Output := SimAuto.GetParameters('bus', FieldBusArray, ValueBusArray);
  if (string(Output[0]) <> '') then
    StatusBar1.Panels[1].Text := 'Error: ' + string(Output[0])
  else
    begin
      StatusBar1.Panels[1].Text := 'Parameters got.';
      Memo1.Lines.Add('== GetParameters ==');
      Memo1.Lines.Add('Value : Output[1][i]');
      for i := VarArrayLowBound(Output[1],1) to VarArrayHighBound(Output[1],1) do begin
```

```
Memo1.Lines.Add(FieldBusArray[i] + ' : ' + string(Output[1][i]));  
end;  
Memo1.Lines.Add("");  
end;  
end;
```



---

**GetParametersSingleElement Function: Sample Code Matlab®**


---

```
% This example loads all buses in the case, and then gets
% some parameters of the last bus in the list

% validcase is a global variable – check case is open
if validcase
    % Gets all buses in the case
    output = simauto.ListOfDevices('bus','');
    if ~(strcmp(output{1},''))
        disp(output{1})
        validbusarray = false;
    else
        % Puts the buses in row vector busarray
        for i=size(output{2}{1},1):size(output{2}{1},2)
            busarray(i,1) = output{2}{1}(i);
        end
        disp('Successful ListOfDevices')
        disp(busarray)
        validbusarray = true;
    end
end

% validbusarray is a global variable – check buses are loaded
if validcase & validbusarray
    % Gets parameters for last bus of busarray
    fieldarray = {'pwbusnum' 'pwbusname' 'pwbusvolt' 'pwbusangle'};
    valuearray = [busarray(size(busarray,1)) '0' '0' '0'];
    valuelist = num2cell(valuearray);
    output = simauto.GetParametersSingleElement('bus',fieldarray,valuelist);
    if ~(strcmp(output{1},''))
        disp(output{1})
    else
        % Puts the buses in row vector busparam
        paramlist = transpose(output{2});
        for i=size(paramlist,1):size(paramlist,2)
            busparam(i,1) = paramlist(i);
        end
        disp('Successful GetParameters for Bus')
        disp(fieldarray)
        disp(busparam)
    end
end
```

---

**GetParametersSingleElement Function: Sample Code Microsoft® Visual Basic for Excel**


---

```

Private Sub btnGetParametersSingleElement_Click()
Dim objtype, filter As String
Dim xlWB As Excel.Workbook
Set xlApp = Excel.Application

' Checks connection and open case
' SimAuto and caseopen are global variables
If Not SimAuto Is Nothing And caseopen Then
objtype = "bus"
Dim fieldArray As Variant
fieldArray = Array("pwBusNum", "pwBusName", "pwBusKVolt", _
"pwBusPUVolt", "pwBusAngle")
Dim ValueArray As Variant
ValueArray = Array(1, 0, 0, 0, 0)
output = SimAuto.GetParameters(objtype, fieldArray, ValueArray)
If output(0) <> "" Then
DisplayErrorMessage output(0)
Else
DisplayMessage "Succesful GetParametersSingleElement"
' Prepares additional worksheet
Set xlWB = xlApp.Workbooks.Add
' Copies list of devices in worksheet
With xlWB
Sheets("sheet2").Activate
Sheets("sheet2").Name = "GetParametersSingleElement"
With Sheets("GetParametersSingleElement")
Dim i As Integer
Range(Cells(1, 5), Cells(200, 7)).Clear
Cells(1, 1) = "List of Devices for " + objtype + ":"
' Setup fields as subheader
For i = LBound(fieldArray) To UBound(fieldArray)
Cells(2, i + 1) = fieldArray(i)
Next i
' Determine number of fields retrieved
Dim lowfld, highfld As Integer
lowfld = LBound(output(1), 1)
highfld = UBound(output(1), 1)
DisplayMessage "Number of Fields: " + Str(lowfld) + Str(highfld)
For i = lowfld To highfld
Cells(j + 3, i + 1) = output(1)(i)
Next i
End With
End With
End If
End If
End Sub

```



---

## GetParametersMultipleElement Function

---

The GetParametersMultipleElement function is used to request the values of specified fields for a set of objects in the load flow case. The function can return values for all devices of a particular type, or can return values for only a list of devices of a particular type based on an [advanced filter](#) defined for the loaded case.

### Function Prototype

**GetParametersMultipleElement(ObjectType, ParamList, FilterName)**

### Parameter Definitions

**ObjectType : String** The type of object you are changing parameters for.

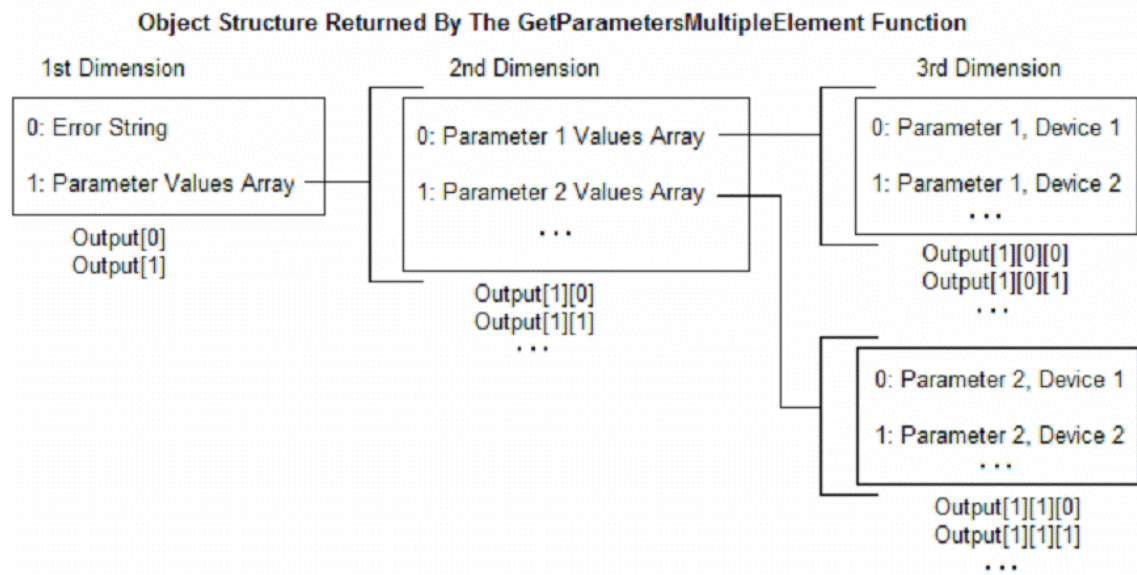
**ParamList : Variant** A variant array storing strings. This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables for the specific device, or the device cannot be identified. The remaining field variables in the array define which values to retrieve from Simulator.

**FilterName : String** The name of an [advanced filter](#) defined in the load flow case open in the Simulator Automation Server. If no filter is desired, then simply pass an empty string. If a filter name is passed but the filter cannot be found in the loaded case, the server will default to returning all objects in the case of type ObjType.

### Output

GetParametersMultipleElement returns a set of nested arrays containing the parameter values for the device type requested. The number of arrays of values returned depends on the number of fields in ParamList.

The Output structure of GetParametersMultipleElement is shown in the following figure.



As you can see, to access the first parameter value for the first device, `Output[1][0][0]` would be the correct array index. For example, the bus number for the first bus would be stored at `Output[1][0][0]` after calling

`Output = GetParametersMultipleElement('Bus', fieldarray, '')`, and assuming that we have `fieldarray = Array(pwBusnum, pwBusName)`.

---

**GetParametersMultipleElement Sample Code Borland® Delphi**


---

```
// This example retrieves some parameters for all buses of the
// loaded case, using the GetParametersMultipleElement function

procedure TMainForm.RunGPMEClick(Sender: TObject);
var
  FieldBusArray : OLEVariant;
  i,j : Integer;
begin
  // Declares fields array to be sent to Excel
  FieldBusArray := VarArrayCreate([1,5], varOleStr);
  FieldBusArray[1] := 'pwBusNum';
  FieldBusArray[2] := 'pwBusname';
  FieldBusArray[3] := 'pwBusKVolt';
  FieldBusArray[4] := 'pwBusPUVolt';
  FieldBusArray[5] := 'pwBusAngle';

  // Gets parameters with Multiple Element function
  Output := SimAuto.GetParametersMultipleElement('bus', FieldBusArray, "");
  if (string(Output[0]) <> "") then
    StatusBar1.Panels[1].Text := 'Error: ' + string(Output[0])
  else
    begin
      StatusBar1.Panels[1].Text := 'Parameters got.';
      Memo1.Lines.Add('== GetParametersMultipleElement ==');
      Memo1.Lines.Add('Value : Output[1][i][j]');
      for i := VarArrayLowBound(Output[1],1) to VarArrayHighBound(Output[1],1) do begin
        for j := VarArrayLowBound(Output[1][i],1) to VarArrayHighBound(Output[1][i],1)
        do begin
          Memo1.Lines.Add(FieldBusArray[i] + '(' + IntToStr(j) + ') : ' +
            string(Output[1][i][j]));
        end;
      end;
      Memo1.Lines.Add("");
    end;
end;
```

---

**GetParametersMultipleElement Function: Sample Code Matlab®**


---

```
% This example loads all buses in the case, and then gets
% some parameters of such buses

% validcase is a global variable – check case is open
if validcase
    % Gets all buses in the case
    output = simauto.ListOfDevices('bus', '');
    if ~(strcmp(output{1}, ''))
        disp(output{1})
        validbusarray = false;
    else
        % Puts the buses in row vector busarray
        for i=size(output{2}{1},1):size(output{2}{1},2)
            busarray(i,1) = output{2}{1}(i);
        end
        disp('Successful ListOfDevices')
        disp(busarray)
        validbusarray = true;
    end
end

% validbusarray is a global variable – check buses are loaded
if validcase & validbusarray
    % Gets parameters for all buses
    fieldarray = {'pwbusnum' 'pwbusname' 'pwbusvolt' 'pwbusangle'};
    output = simauto.GetParametersMultipleElement('bus', fieldarray, ' ');
    if ~(strcmp(output{1}, ''))
        disp(output{1})
    else
        % Puts the buses in matrix busesparam
        paramlist = transpose(output{2});
        for i=size(paramlist,1):size(paramlist,2)
            for j=size(paramlist{i},2):size(paramlist{i},1)
                busesparam(j,i) = paramlist{i}(j);
            end
        end
        disp('Successful GetParametersMultipleElement')
        disp(fieldarray)
        disp(busesparam)
    end
end
```

---

**GetParametersMultipleElement Function: Sample Code Microsoft® Visual Basic for Excel**


---

```

Private Sub btnGetParametersMultiple_Click()
Dim objtype, filter As String
Dim xlWB As Excel.Workbook
Set xlApp = Excel.Application

' Checks connection and open case
' SimAuto and caseopen are global variables
If Not SimAuto Is Nothing And caseopen Then
objtype = "bus"
filter = ""
Dim fieldArray As Variant
fieldArray = Array("pwBusNum", "pwBusName", "pwBusKVolt", _
"pwBusPUVolt", "pwBusAngle")
output = SimAuto.GetParametersMultipleElement(objtype, fieldArray, filter)
If output(0) <> "" Then
DisplayErrorMessage output(0)
Else
DisplayMessage "Succesful GetParametersSingleElement"
' Prepares additional worksheet
Set xlWB = xlApp.Workbooks.Add
' Copies list of devices in worksheet
With xlWB
Sheets("sheet2").Activate
Sheets("sheet2").Name = "GetParametersSingleElement"
With Sheets("GetParametersSingleElement")
Dim i, j As Integer
Range(Cells(1, 5), Cells(200, 7)).Clear
Cells(1, 1) = "List of Devices for " + objtype + ":"
' Setup fields as subheader
For i = LBound(fieldArray) To UBound(fieldArray)
Cells(2, i + 1) = fieldArray(i)
Next i
' Determine number of fields retrieved
Dim lowfld, highfld As Integer
lowfld = LBound(output(1), 1)
highfld = UBound(output(1), 1)
' Determine number of objects retrieved
Dim lowobj, highobj As Integer
lowobj = LBound(output(1)(lowkeyf), 1)
highobj = UBound(output(1)(lowkeyf), 1)
DisplayMessage "Number of Fields: " + Str(lowfld) + Str(highfld)
DisplayMessage "Number of objects: " + Str(lowobj) + Str(highobj)
For i = lowfld To highfld
For j = lowobj To highobj
Cells(j + 3, i + 1) = output(1)(i)(j)
Next j
Next i

```



End With  
End With  
End If  
End If  
End Sub

---

**GetParametersMultipleElementFlatOutput Function**

---

This function operates the same as the [GetParametersMultipleElement](#) function, only with one notable difference. The values returned as the output of the function are returned in a single-dimensional vector array, instead of the multi-dimensional array as described in the [GetParametersMultipleElement](#) topic. The function returns the parameter values for the device type requested, and the number of values returned depends on the ParamList and any [advanced filter](#) that impacts the number of devices returned.

The format of the output array is the following:

[errorString, NumberOfObjectsReturned, NumberOfFieldsPerObject, Ob1Fld1, Ob1Fld2, ..., Ob(n)Fld(m-1), Ob(n)Fld(m)]

The data is thus returned in a single dimension array, where the parameters NumberOfObjectsReturned and NumberOfFieldsPerObject tell you how the rest of the array is populated. Following the NumberOfObjectsReturned parameter is the start of the data. The data is listed as all fields for object 1, then all fields for object 2, and so on. You can parse the array using the NumberOf... parameters for objects and fields.

---

**GetParameters Function**

---

This function is maintained in versions of Simulator later than version 9 for compatibility with Simulator version 9. This function is replaced by [GetParametersSingleElement](#). See also [GetParametersMultipleElement](#).

---

## ListOfDevices Function

---

The ListOfDevices function is used to request a list of objects and their [key fields](#) from the [Simulator Automation Server](#). The function can return all devices of a particular type, or can return only a list of devices of a particular type based on an [advanced filter](#) defined for the loaded case. This function is best used in conjunction with a looping procedure and the [ChangeParameters](#) or [GetParametersSingleElement](#) functions to process a group of devices.

### Function Prototype

**ListOfDevices(ObjType, filterName)**

### Parameter Definitions

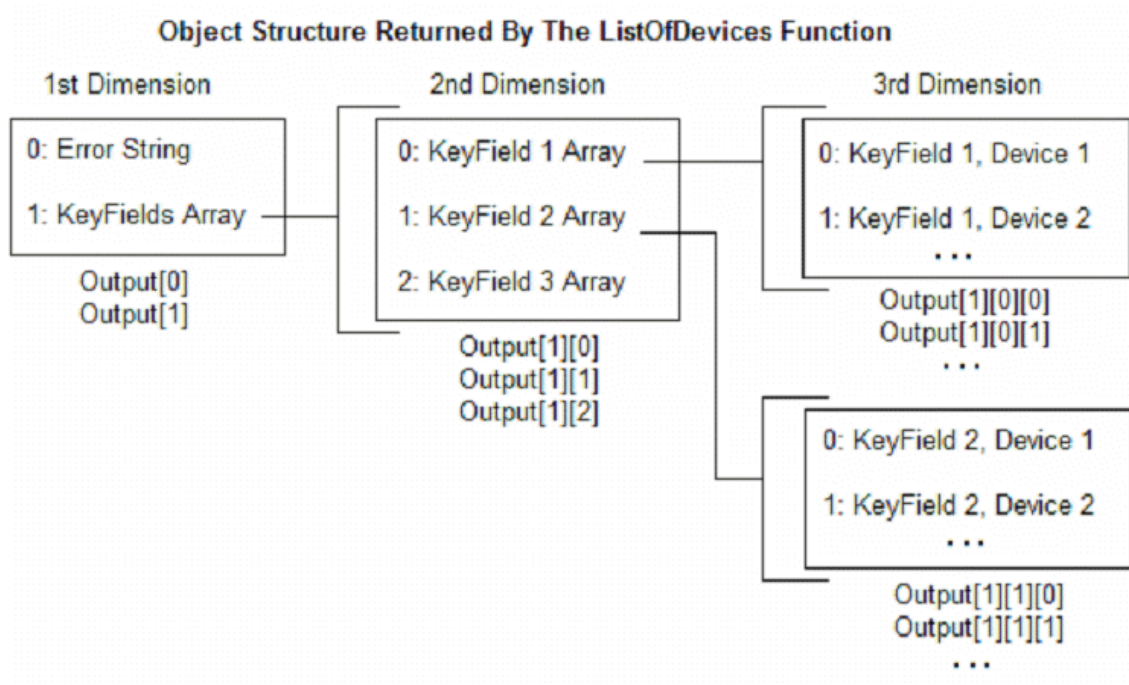
**ObjType : String** The type of object for which you are acquiring the list of devices.

**FilterName : String** The name of an [advanced filter](#) defined in the load flow case open in the Simulator Automation Server. If no filter is desired, then simply pass an empty string. If the filter cannot be found, the server will default to returning all objects in the case of type ObjType.

### Output

ListOfDevices returns a set of nested arrays containing the [key field](#) values for the device type requested. The number of arrays of values returned depends on the object type selected. For instance, buses have only one key field (the bus number) so calling ListOfDevices for buses will return only one array of values—the bus numbers. On the other hand, calling ListOfDevices for branches will return three arrays of values—the "From" bus, "To" bus, and ID—for each branch in the case meeting the specified filter.

The arrays containing the key field values for each device are arranged as shown in the following figure.



As you can see, to access the first key field value for the first device, `Output[1][0][0]` would be the correct array index. For example, the bus number (which is the bus key field) for the first bus would be stored at `Output[1][0][0]` after calling `Output = ListOfDevices('Bus', '')`.

One unique limitation of the `ListOfDevices` function from other `SimAuto` functions is that this is the only function that returns the output as strongly typed variables. The bus numbers are always returned as Long Integers, and the Circuit ID values are returned as strings. This was actually an oversight during the design of `SimAuto`. In all other `SimAuto` functions, the values are returned as Variant types, with each value within the variant being a string. This was the intended operation for this function as well. Since the Automation Server interface was released with the errant inclusion of the `ListOfDevices` function, it could not be modified. Therefore, another function, `ListOfDevicesAsVariantStrings`, has been created. This function returns all values in variant variables, with each as a string within the variant type.

---

**ListOfDevices Function: Sample Code for Borland® Delphi**


---

**Sample Code**

```
// Runs Available Transfer Capability Routine
// Executes ATC Calculations among all areas
// and sends results to Excel
procedure TMainForm.RunATCClick(Sender: TObject);
var
  i, j, LowB, HighB : Integer;
  ValuesAreaArray : OLEVariant;
begin
  // Obtain all the areas
  Output := SimAuto.ListOfDevices('area', "");
  if (string(Output[0]) <> "") then
    StatusBar1.Panels[1].Text := 'Error: ' + string(Output[0])
  else
    begin
      ValuesAreaArray := Output[1][0];
      LowB := VarArrayLowBound(ValuesAreaArray, 1);
      HighB := VarArrayHighBound(ValuesAreaArray, 1);
      // Executes loop
      for i := LowB to HighB do
        for j := LowB to HighB do begin
          if (i <> j) then begin
            // Runs ATC calculations
            OutputATC := SimAuto.RunScriptCommand('entermode(atc); ' +
              'atcdetermine([Area ' + IntToStr(ValuesAreaArray[i]) +
              '], [Area ' + IntToStr(ValuesAreaArray[j]) + '])');
            if (string(OutputATC[0]) <> "") then
              StatusBar1.Panels[1].Text := 'Error: ' + string(OutputATC[0])
            else begin
              // Sends ATC results to Excel
              Output := SimAuto.SendToExcel('transferlimiter', "", 'all');
              if (string(Output[0]) <> "") then
                StatusBar1.Panels[1].Text := 'Error: ' + string(Output[0])
              else
                StatusBar1.Panels[1].Text := 'ATC calculations done.';
            end;
          end;
        end;
      end;
    end;
end;
```

---

**ListOfDevices Function: Sample Code for Matlab®**


---

**Sample Code**

```
%A list of branches is desired, without using any filter
DeviceType = 'Branch';
FilterName = '';
%Execute the ListOfDevices command
Output = SimAuto.ListOfDevices(DeviceType,FilterName);
%If the first cell in SimAutoOutput ~= "", then that means an error
%occurred.
if ~(strcmp(SimAutoOutput{1},''))
    disp(SimAutoOutput{1})
else
    %Otherwise, no errors. Display the branch information.
    disp('ListOfDevices successful')

    %Devicelist1 contains the From: bus number
    %Devicelist2 contains the To: bus number
    %Devicelist3 contains the Bus identifier
    devicelist1 = double(transpose(SimAutoOutput{2}{1}));
    devicelist2 = double(transpose(SimAutoOutput{2}{2}));
    devicelist3 = SimAutoOutput{2}{3};

    %If the device list is greater than 25, don't bother attempting to
    %display it on the screen.
    if (size(devicelist1,1) > 25)
        disp('Device list exceeds 25; use "devicelist" to manage list of devices')
    else
        %Otherwise, display the branches' information.
        disp(DeviceType)
        disp('From/To/Identifier')
        for counter = 1:size(devicelist1,1)
            %num2str converts the numbers within devicelist1 and
            %devicelist2 to strings for output with disp(). the char()
            %function is called on devicelist3's members because the
            %SimAuto object returns a character array (as opposed to a
            %properly Matlab-format string) and this array must be
            %converted to a Matlab-format string.
            disp([num2str(devicelist1(counter)) ' ' ...
                num2str(devicelist2(counter)) ' ' ...
                char(devicelist3(counter))])
        end
    end
end
```

---

**ListOfDevices Function: Sample Code for Microsoft® Visual Basic for Excel**


---

**Sample Code**

```

Private Sub DisplayMessage(ByVal SentText As String)
    TextBox.Text = TextBox.Text + SentText + vbCrLf + vbCrLf
End Sub

Private Sub btnListOfDevices_Click()
    Dim objtype, filter As String
    Dim xlWB As Excel.Workbook
    Set xlApp = Excel.Application

    ' Checks connection and open case
    ' SimAuto and caseopen are global variables
    If Not SimAuto Is Nothing And caseopen Then
        objtype = "branch"
        filter = ""
        output = SimAuto.ListOfDevices(objtype, filter)
        If output(0) <> "" Then
            DisplayMessage output(0)
        Else
            DisplayMessage "Successful List Of Devices"
        ' Prepares additional worksheet
        Set xlWB = xlApp.Workbooks.Add
        ' Copies list of devices in worksheet
        With xlWB
            Sheets("sheet1").Activate
            Sheets("sheet1").Name = "ListOfDevices"
            With Sheets("ListOfDevices")
                Dim i, j As Integer
                Range(Cells(1, 5), Cells(200, 7)).Clear
                Cells(1, 1) = "List of Devices for " + objtype + ":"
                Cells(2, 1) = "From Bus Num"
                Cells(2, 2) = "To Bus Num"
                Cells(2, 3) = "ID"
                ' Determine number of key fields retrieved
                Dim lowkeyf, highkeyf As Integer
                lowkeyf = LBound(output(1), 1)
                highkeyf = UBound(output(1), 1)
                DisplayMessage "Number of Key Fields: " + Str(lowkeyf) + Str(highkeyf)
                ' Determine number of objects retrieved
                Dim lowobj, highobj As Integer
                lowobj = LBound(output(1)(lowkeyf), 1)
                highobj = UBound(output(1)(lowkeyf), 1)
                DisplayMessage "Number of objects: " + Str(lowobj) + Str(highobj)
                For i = lowkeyf To highkeyf
                    For j = lowobj To highobj
                        Cells(j + 3, i + 1) = output(1)(i)(j)
                    Next j
                
```



```
Next i
End With
End With
End If
End If
End Sub
```

---

### ListOfDevicesAsVariantStrings Function

---

This function operates the same as the [ListOfDevices](#) function, only with one notable difference. The values returned as the output of the function are returned as Variants of type String. The ListOfDevices function was errantly released returning the values strongly typed as Integers and Strings directly, whereas all other SimAuto functions returned data as Variants of type String. This function was added to also return the data in the same manner. This solved some compatibility issues with some software languages.

---

**ListOfDevicesFlatOutput Function**

---

This function operates the same as the [ListOfDevices](#) function, only with one notable difference. The values returned as the output of the function are returned in a single-dimensional vector array, instead of the multi-dimensional array as described in the ListOfDevices topic. The function returns the key field values for the device, typically in the order of bus number 1, bus number 2 (where applicable), and circuit identifier (where applicable). These are the most common key fields, but some object types do have other key fields as well.

The format of the output array is the following:

[errorString, NumberOfObjectsReturned, NumberOfFieldsPerObject, Ob1Fld1, Ob1Fld2, ..., Ob(n)Fld(m-1), Ob(n)Fld(m)]

The data is thus returned in a single dimension array, where the parameters NumberOfObjectsReturned and NumberOfFieldsPerObject tell you how the rest of the array is populated. Following the NumberOfObjectsReturned parameter is the start of the data. The data is listed as all fields for object 1, then all fields for object 2, and so on. You can parse the array using the NumberOf... parameters for objects and fields.

---

## LoadState Function

---

LoadState is used to load the system state previously saved with the [SaveState](#) function. Note that LoadState will not properly function if the system topology has changed due to the addition or removal of the system elements.

### Function Prototype

**LoadState()**

### Parameter Definitions

No parameters are passed.

### Output

LoadState returns only one element in Output—any errors which may have occurred when attempting to execute the function.

---

**LoadState Function: Sample Code**

---

**Microsoft® Visual Basic for Applications**

' Make the LoadState call  
Output = SimAuto.LoadState()

**Matlab®**

% Make the LoadState call  
Output = SimAuto.LoadState();

---

## OpenCase Function

---

The OpenCase function will load a PowerWorld® Simulator load flow file into the [Simulator Automation Server](#). This is equivalent to opening a file using the **File > Open Case** menu option in Simulator.

### Function Prototype

**OpenCase(FileName)**

### Parameter Definitions

**FileName : String** The name of the PowerWorld® Simulator case file to be loaded into the Simulator Automation Server. This string includes the directory location and full file name.

### Output

OpenCase returns only one element in Output—if the file cannot be found or an error occurs while reading the file.

---

**OpenCase Function: Sample Code**


---

**Borland® Delphi**

```

Output := SimAuto.OpenCase('c:\simauto\examples\b7opf.pwb');
if (string(Output[0]) <> '') then
  StatusBar1.Panels[1].Text := 'Error: ' + string(Output[0]);
else
begin
  StatusBar1.Panels[1].Text := 'Open Case successful.';
  // Perform activities with opened case
end;

```

**Microsoft® Visual Basic for Applications**

```

Output = SimAuto.OpenCase("c:\simauto\examples\b7opf.pwb")
If output(0) <> "" Then
  MsgBox(output(0))
Else
  ' Perform activities with the opened case
End If

```

**Matlab®**

```

Output = SimAuto.OpenCase('c:\simauto\examples\b7opf.pwb')
%If the first cell in Output ~= "", then that means an error
%occurred.
if ~(strcmp(Output{1},""))
  disp(Output{1})
else
  %Otherwise, no errors. Perform activities.
  disp('Open Case successful')
end

```

---

## ProcessAuxFile Function

---

The ProcessAuxFile function will load a PowerWorld® Auxiliary file into the [Simulator Automation Server](#). This allows you to create a text file (conforming to the PowerWorld® Auxiliary file format) that can list a set of data changes and other information for making batch changes in Simulator

### Function Prototype

**ProcessAuxFile(FileName)**

### Parameter Definitions

**FileName : String** The name of the PowerWorld® Auxiliary file to be loaded into the Simulator Automation Server. This string includes the directory location and full file name.

### Output

ProcessAuxFile returns only one element in Output—any errors which may have occurred when attempting to load the file.



---

**ProcessAuxFile Function: Sample Code**

---

**Microsoft® Visual Basic for Applications**

Dim filename As String

' Setup name of aux file to run

filename = "c:\b7opf\_ctglist.aux"

' Make the processAuxFile call

Output = SimAuto.ProcessAuxFile(filename)

**Matlab®**

% Setup name of aux file to run

filename = 'c:\b7opf\_ctglist.aux';

% Make the processAuxFile call

Output = SimAuto.ProcessAuxFile(filename);

---

## RunScriptCommand Function

---

The RunScriptCommand function is used to execute a list of [script statements](#). The [script actions](#) are those included in the script sections of the [Auxiliary Files](#). If an error occurs trying to run a script command, an error will be returned through EString.

### Function Prototype

**RunScriptCommand(Statements)**

### Parameter Definitions

**Statements : String** The block of script actions to be executed. Each script statement must end in a semicolon. The block of script actions should **not** be enclosed in curly braces.

### Output

RunScriptCommand returns only one element in Output—any errors which may have occurred when attempting to load or run the auxiliary file.

---

**RunScriptCommand Function: Sample Code**

---

**Microsoft® Visual Basic for Applications**

Dim scriptcommand As String

' Set script command to cause Simulator to enter Run Mode  
scriptcommand = "EnterMode(RUN)"

' Make the RunScriptCommand call  
Output = SimAuto.RunScriptCommand(scriptcommand);

' Set script command to cause Simulator to perform a single,  
' standard solution  
scriptcommand = "SolvePowerFlow(RECTNEWT)"

' Make the RunScriptCommand call  
Output = SimAuto.RunScriptCommand(scriptcommand);

**Matlab®**

% Set script command to cause Simulator to enter Run Mode  
scriptcommand = 'EnterMode(RUN)';

% Make the RunScriptCommand call  
Output = SimAuto.RunScriptCommand(scriptcommand);

% Set script command to cause Simulator to perform a single,  
% standard solution  
scriptcommand = 'SolvePowerFlow(RECTNEWT)';

% Make the RunScriptCommand call  
Output = SimAuto.RunScriptCommand(scriptcommand);

---

## SaveCase Function

---

The SaveCase function is used to save a case previously loaded in the [Simulator Automation Server](#) using the [OpenCase](#) function. The function allows you to specify a file name and a format for the save file.

### Function Prototype

**SaveCase(FileName, EString, FileType, Overwrite)**

### Parameter Definitions

**FileName : String** The name of the file you wish to save as, including file path.

**FileType : String** A string indicating the format of the written case file. An empty string will return an error. The following list is the currently supported list of string identifiers and the file types they represent.

"PTI23" PTI version 23 (raw)

"PTI24" PTI version 24 (raw)

"PTI25" PTI version 25 (raw)

"PTI26" PTI version 26 (raw)

"PTI27" PTI version 27/28 (raw)

"PTI29" PTI version 29 (raw)

"PTI30" PTI version 30 (raw)

"GE" GE PSLF (epc)

"IEEE" IEEE common format (cf)

"PWB70" PowerWorld Binary version 7.0 (pwb)

"PWB" PowerWorld Binary (most recent) (pwb)

**Overwrite : Boolean** A Boolean value which indicates whether to overwrite a file if FileName already exists. If Overwrite is set to false and the file specified by FileName already exists, SaveCase will return an error message and do nothing to the file.

### Output

SaveCase returns only one element in Output—any errors which may have occurred when attempting to save the case.

---

**SaveCase Function: Sample Code**

---

**Microsoft® Visual Basic for Applications**

' Save the case as a PWB file

Output = SimAuto.SaveCase("c:\b7opfcopy.pwb", "PWB", true)

' Save the case as a PTI file

Output = SimAuto.SaveCase("c:\b7opfcopy.raw", "PTI", true)

**Matlab®**

% Setup name of PWB file to write

filenamepwb = 'c:\b7opfcopy.pwb';

% Setup name of PTI file to write

filenamepti = 'c:\b7opfcopy.raw';

% Make the SaveCase call for the PWB file

Output = SimAuto.SaveCase(filenamepwb, 'PWB', true);

% Make the SaveCase call for the PTI file

Output = SimAuto.SaveCase(filenamepti, 'PWB', true);

---

## SaveState Function

---

SaveState is used to save the current state of the power system. This can be useful if you are interested in comparing various cases, much as the [Difference Flows](#) feature works in the Simulator application.

### Function Prototype

**SaveState()**

### Parameter Definitions

No parameters are passed.

### Output

SaveState returns only one element in Output—any errors which may have occurred when attempting to execute the function.

---

**SaveState Function: Sample Code**

---

**Microsoft® Visual Basic for Applications**

' Make the SaveState call

Output = SimAuto.SaveState()

**Matlab®**

% Make the SaveState call

Output = SimAuto.SaveState();

---

## SendToExcel Function

---

The SendToExcel function can be called to send data from the [Simulator Automation Server](#) to an Excel spreadsheet. The function is flexible in that you can specify the type of object data you want to export, an [advanced filter](#) name for a filter you want to use, and as many or as few [field types](#) as desired that are supported by the type of object. The first time this function is called, a new instance of Excel will be started, and the data requested will be pasted to a new sheet. For each subsequent call of this function, the requested data will be pasted to a new sheet within the same workbook, until the workbook is closed.

### Function Prototype

**SendToExcel(ObjectType, FilterName, FieldList)**

### Parameter Definitions

**ObjectType : String** A string describing the type of object for which you are requesting data.

**FilterName : String** The name of an [advanced filter](#) which was previously defined in the case before being loaded in the Simulator Automation Server. If no filter is desired, then simply pass an empty string. If a filter name is passed but the filter cannot be found in the loaded case, no filter is used.

**FieldList : Variant** This parameter must either be an array of fields for the given object or the string "all". As an array, FieldList contains an array of strings, where each string represents an object field variable, as defined in the section on [PowerWorld Object Variables](#). If, instead of an array of strings, the single string "all" is passed, the Simulator Automation Server will use predefined default fields when exporting the data.

### Output

SendToExcel returns only one element in Output—any errors which may have occurred when attempting to execute the function.



---

**SendToExcel Function: Sample Code**


---

**Microsoft® Visual Basic for Applications**

Dim FieldList As Variant

' Setup fieldlist to send the bus number, gen id and gen agc to Excel

FieldList = Array("pwBusNum", "pwGenID", "pwGenAGCable")

' Make the SendToExcel call

' By specifying the parameter FieldList, only the three fields

' for each generator will be returned

Output = SimAuto.SendToExcel("gen", "", "FieldList")

' Sending the string "all" instead of a fieldlist array

' writes all predefined fields to the Excel spreadsheet

Output = SimAuto.SendToExcel("gen", "", "all")

Note: This function call will send the values of the fields in FieldList to an Excel workbook for all the generators in the load flow case. If a filter name had been passed instead of an empty string, Simulator would have located and used a pre-defined advanced filter and applied it to the information if it was found.

**Matlab®**

% Setup fieldlist to send the bus number, gen id and gen agc to Excel

fieldlist = {'pwBusNum' 'pwGenID' 'pwGenAGCable' };

% Make the SendToExcel call

Output = SimAuto.SendToExcel('gen', " , FieldList);

% Sending the string 'all' instead of a fieldlist array

% writes all predefined fields to the Excel spreadsheet

Output = SimAuto.SendToExcel('gen', " , 'all');

Note: This function call will send the values of the fields in FieldList to an Excel workbook for all the generators in the load flow case. If a filter name had been passed instead of an empty string, Simulator would have located and used a pre-defined advanced filter and applied it to the information if it was found.

---

## WriteAuxFile Function

---

The WriteAuxFile function can be used to write data from the case in the [Simulator Automation Server](#) to a PowerWorld® Auxiliary file. The function is flexible in that you can specify the type of object data you want to export, an [advanced filter](#) name for a filter you want to use, and as many or as few field types as desired that are supported by the type of object. In addition, you can specify a new file name for each call to WriteAuxFile, or you can specify the same file name and append the data to the file.

### Function Prototype

**WriteAuxFile(FileName, FilterName, ObjectType, EString, ToAppend, FieldList)**

### Parameter Definitions

**FileName : String** The name of the PowerWorld® Auxiliary file you wish to save.

**FilterName : String** The name of an [advanced filter](#) which was previously defined in the case before being loaded in the Simulator Automation Server. If no filter is desired, then simply pass an empty string. If a filter name is passed but the filter cannot be found in the loaded case, no filter is used.

**ObjectType : String** A string describing the type of object for which you are requesting data.

**ToAppend : Boolean** If you have given a file name of an auxiliary file that already exists, then the file will either be appended to or overwritten according to the setting of this parameter. If ToAppend is *False* and the file already exists, WriteAuxFile will return an error message and do nothing to the file.

**FieldList : Variant** This parameter must either be an array of fields for the given object or the string "all". As an array, FieldList contains an array of strings, where each string represents an object field variable, as defined in the section on [PowerWorld Object Variables](#). If, instead of an array of strings, the single string "all" is passed, the Simulator Automation Server will use predefined default fields when exporting the data.

### Output

WriteAuxFile returns only one element in Output—any errors which may have occurred when attempting to execute the function.

---

**WriteAuxFile Function: Sample Code**


---

**Microsoft® Visual Basic for Applications**

```
Dim FieldList As Variant
```

```
Dim auxfilename As String
```

```
' Setup FieldList to send the bus number, gen id and gen agc
```

```
FieldList = Array("pwBusNum", "pwGenID", "pwGenAGCable")
```

```
' Aux file to write to
```

```
auxfilename = "c:\businfo.aux"
```

```
' Make the WriteAuxFile call
```

```
' By specifying the parameter FieldList, only the three fields
```

```
' for each generator will be returned
```

```
Output = SimAuto.WriteAuxFile(auxfilename, "", "gen", true, FieldList)
```

```
' Sending the string "all" instead of the FieldList array
```

```
' writes all predefined fields to the Excel spreadsheet
```

```
Output = SimAuto.SendToExcel(auxfilename, "", "gen", true, "all")
```

Note: This function call will send the values of the fields in FieldList to an auxiliary file for all the generators in the load flow case. If a filter name had been passed instead of an empty string, Simulator would have located and used a pre-defined advanced filter and applied it to the information if it was found.

**Matlab®**

```
% Setup FieldList to send the bus number, gen id and gen agc
```

```
fieldlist = {'pwBusNum' 'pwGenID' 'pwGenAGCable' };
```

```
% Aux file to write to
```

```
auxfilename = 'c:\businfo.aux';
```

```
% Make the WriteAuxFile call
```

```
Output = SimAuto.WriteAuxFile(auxfilename, "", 'gen', true, FieldList);
```

```
% Sending the string 'all' instead of the FieldList array
```

```
% writes all predefined fields to the .aux file
```

```
Output = SimAuto.WriteAuxFile(auxfilename, "", 'gen', true, 'all');
```

Note: This function call will send the values of the fields in FieldList to an auxiliary file for all the generators in the load flow case. If a filter name had been passed instead of an empty

string, Simulator would have located and used a pre-defined advanced filter and applied it to the information if it was found.

---

**Simulator Automation Server Properties**

---

The following list of parameters is currently available once the SimulatorAuto object is set in your code. Check the help sections on these properties to see more detail on the particular property.

[ExcelApp](#)

[CurrentDir](#)

[ProcessID](#)

[RequestBuildDate](#)

---

## ExcelApp Property

---

The [Simulator Automation Server](#) has the ability to send data from Simulator to an Excel spreadsheet using the [SendToExcel](#). By default, the Simulator Automation Server starts an instance of Excel the first time one of the above functions is called. Each subsequent call to these two functions will then send data to the same instance of Excel, until it is manually closed by the user. The ExcelApp property allows the user to gain access to the instance of Excel used by the Simulator Automation Server from within its own code. Thus the user can write code to manipulate the external instance of Excel. In addition, the ExcelApp property can be set by the user's code, meaning that the user can initialize an external instance of Excel from within their own code and set the ExcelApp property to their external instance of Excel. Simulator itself is limited to starting only one instance of Excel on its own, but with the ExcelApp property allowing you to set the instance of Excel that Simulator uses on the fly, you can generate multiple instances of Excel within your code, and handle setting the ExcelApp property of the Simulator Automation server to the desired Excel instance depending on the data you want to send to Excel.

The ExcelApp property is a variable of type Variant that returns the pointer to an object representing the external instance of Excel.

### ExcelApp : Variant

To gain access to the external instance of Excel stored in the ExcelApp property, you first need to initialize a variable as an object. The following are a couple of examples in Borland Delphi and Microsoft Visual Basic.

### Examples

#### Borland Delphi 5

- **Var MyExc : TObject**
- **MyExc = SimAuto.ExcelApp***{Makes the connection to the external instance}*
- *{Perform activities with the Excel instance}*
- **MyExc.Quit***{Closes the external instance if called; do not call if you wish the instance to remain open}*
- **MyExc.Free***{Removes the connection to the external instance}*

#### Microsoft Visual Basic

- **Dim MyExc As Variant**
- **Set MyExc = SimAuto.ExcelApp***{Makes the connection to the external instance}*
- *{Perform activities with the Excel instance}*
- **MyExc.Quit***{Closes the external instance if called; do not call if you wish the instance to remain open}*
- **Set MyExc = Nothing***{Removes the connection to the external instance}*

---

**ExcelApp Property: Sample Code**

---

**Microsoft® Visual Basic for Applications**

```
Dim ExcelObject As Variant
If Not (IsEmpty(SimAuto.ExcelApp)) Then
    Set ExcelObject = SimAuto.ExcelApp
    ExcelObject.DisplayAlerts = False
    ExcelObject.Quit
Else
    MsgBox("Attempted to obtain Excel COM object from " + _
    "Simulator, but there is not one open.")
End If
```

**Matlab®**

*MATLAB is currently unable to handle COM object properties returning COM objects themselves, such as the ExcelApp property of Simulator.*

---

### **CurrentDir Property**

---

The CurrentDir property of the [Simulator Automation Server](#) allows you to retrieve or set the working directory for the currently running SimulatorAuto process. This is most useful if using relative filenames (e.g. "relativename.aux" versus "C:\Program Files\PowerWorld\Working\abosultename.aux") when specifying files.



---

**CurrentDir Property: Sample Code**

---

**Microsoft® Visual Basic for Applications**

' Display the current directory

MsgBox(SimAuto.CurrentDir)

' Set the current directory to c:\

SimAuto.CurrentDir = "c:\"

**Matlab®**

% Display the current directory

disp(SimAuto.CurrentDir)

% Set the current directory to c:\

SimAuto.CurrentDir = 'c:\';

---

## ProcessID Property

---

The ProcessID property of the [Simulator Automation Server](#) allows you to retrieve the process ID of the currently running SimulatorAuto process, as can also be seen through the Task Manager in Windows® NT 4, 2000 and XP. This information can be useful if a forced shutdown of the SimulatorAuto object is needed, as all calls to the SimulatorAuto object are synchronous. This means the SimulatorAuto object will not be destroyed until all calls, no matter the time of execution, have completed.

---

**ProcessID Property: Sample Code**

---

**Microsoft® Visual Basic for Applications**

```
' Display the process ID  
MsgBox(SimAuto.ProcessID)
```

**Matlab®**

```
% Display the process ID  
disp(num2str(SimAuto.ProcessID))
```

---

**RequestBuildDate Property**

---

The RequestBuildDate property of the [Simulator Automation Server](#) allows you to retrieve the build date of the PowerWorld Simulator executable currently running with the SimulatorAuto process. The property returns an integer value that represents a date. This information is useful for verifying the release version of the executable.

---

## PowerWorld Object Variables

---

The ability to access power system data for different objects through various [Simulator Automation Server Functions](#) is based on variables defined in Simulator that can be referred to as Object Field Variables. Each object (i.e. bus, generator, etc.) can have numerous fields associated with it. Each of these fields, in turn, has a variable associated with the field to enable access to the field for the purpose of acquiring or changing data. For example, the [GetParametersSingleElement](#) function has a parameter called ParamList, which is intended to store a list of Object Field Variables for a particular type of object. When the function is called, the Simulator Automation Server will return the values associated with each particular field variable for the type of object specified. These field variables allow for complete flexibility by the user in specifying as many or as few fields for a particular object when acquiring or changing data.

### Examples of Field Variables

**BusGenMW**

**BusNum**

Simulator has literally hundreds of parameters spanning numerous types of device and option specifications. Rather than list all of the field variables and the value they represent in this help file, we have enabled Simulator to automatically generate a text file containing the field variables and a description of what value the variable represents. PowerWorld Corporation highly recommends that you examine this list. To generate this text file, run PowerWorld Simulator and access the **Help** menu. Choose the option **Export Case Object Fields...** The list of fields can either be saved to a text file or sent to Excel. The list will consist of the field variables, the type of variable (string, integer, etc.), and a description of the value the field variable represents, with key fields for different objects marked with an asterisk. The field variables will also be split into sections based on the type of object they are valid for. Note that the same field variable may be available for more than one object, but that the value represented by the field variable might vary for different objects.

### Examples of Field Variables in Listing

BusGenMW Real 'Bus Gen MW'

\* BusNum Integer 'Bus Number' (\* denotes key field in text file)

ATC\_MaxLimElements Integer 'Max # Limiting Elements'

The last note on the Object Field Variables is that some of the object field variables are reused for more than one value for an object. For example, a transmission line has "from" and "to" buses associated with the line. Rather than have separate field variables for values at each terminal bus, the same field variable is used for both, with a colon followed by a number appended to the variable to signify which bus the value represents. As an example, consider the field variable for bus per unit voltage, which is BusPUVolt. Since there are two buses per line, the "From" bus voltage would be represented as BusPUVolt:0, and the "To" bus voltage would be represented as BusPUVolt:1. The enumeration of the field variables always starts with 0 for the first instance. You may note that in the text file of field values that you don't see any field variables with :0 appended to them. Since we did not want you to have to always append the :0 on all field variables, the default for a field variable with no appended :# is 0. Thus you would only need to be concerned with appending the :# for field variables that require a number greater than 0.



---

**Simulator Automation Server (version 9)**

---

**NOTE: The function calls for version 9 of SimAuto have become obsolete in version 10. The reason for the change was due to problems that arose with the structure of the functions in certain programming languages. The function structures have been modified for version 10 of Simulator. If you have code written that uses version 9 SimAuto function structures, it is fairly straightforward to convert the function calls from version 9 to version 10. Please see the updated section on the Simulator Automation Server for version 10. If you still have questions, please contact PowerWorld Corporation.**

The PowerWorld® Automation Server is only available to customers who have purchased the SimAuto add-on for PowerWorld® Simulator. The PowerWorld Simulator Automation Server is intended for enabling a PowerWorld customer with the ability to access PowerWorld Simulator functionality from within a program written externally by the user. The Simulator Automation Server acts as a COM object, which can be accessed from various different programming languages that have COM compatibility. Examples of programming tools with COM compatibility are Borland® Delphi, Microsoft® Visual C++, and Microsoft® Visual Basic, just to name a few. For more information on COM and Automation servers, see the help for Microsoft Windows.

The Automation Server of Simulator works very well in combination with Simulator [Script Commands](#) and [Auxiliary Files](#). It is beneficial to become familiar with these topics when considering using the Simulator Automation Server.

---

### **Installing Simulator Automation Server (version 9)**

---

Installing the [Simulator Automation Server](#) requires no additional steps beyond installing PowerWorld Simulator as normal. When a version of PowerWorld Simulator containing the Simulator Automation Server is installed on your computer, the install program automatically adds the information needed by the Simulator Automation Server to the registry.



---

## Including Simulator Automation Server Functions (version 9)

---

Before you can access the [functions](#) defined by the [Simulator Automation Server](#) when writing the code for your external program, you must first include the library of functions defined for the Simulator Automation Server. This kind of library is referred to as a Type Library, which describes the available functions in a manner that can be interpreted by different programming languages. Importing a Type Library from another program is usually fairly simple, but the procedure does vary depending on the programming tool you are using. Please see the help for your programming tool of choice on how to import a Type Library or COM functions from another program.

### Examples

The following examples are just a few specific examples for certain programming media. The procedure may be different for other programming media not listed. In addition, a procedure given for a certain type of programming media may be one variation from several possible procedures for accomplishing the same task.

#### Borland Delphi 5

- Install the version of PowerWorld Simulator with the Simulator Automation Server included.
- In Delphi 5, choose Import Type Library... from the Project menu.
- In the list of libraries, search for and choose pwrworld Library.
- If pwrworld Library is not in the list, click Add. Find and choose the Pwrworld.exe file from the PowerWorld Simulator directory, and click Open.
- You should see the class name TSimulatorAuto in the list of Class names.
- Click Install to include the PowerWorld Simulator Type Library.

#### Microsoft Visual Basic

- Install the version of PowerWorld Simulator with the Simulator Automation Server included.
- In VB, choose **References...** from the **Tools** menu. Find and choose pwrworld Library from the list of references.
- If pwrworld Library is not in the list, click Browse. Change the file type to \*.exe, find and choose the Pwrworld.exe file from the PowerWorld Simulator directory, and click Open.
- Click OK to install the Simulator Type Library reference.

#### Microsoft Visual C++

- Install the version of PowerWorld Simulator with the Simulation Automation Server included.
- Add **#import "...\\powerworld.exe"** in your external program code, using the full path to the PowerWorld Simulator executable program.
- Add **using namespace pwrworld** in your external program code.

---

## Connecting to Simulator Automation Server (version 9)

---

Once the Type Library or COM functions have been included in your programming environment, the [Simulator Automation Server](#) can be handled as any other object in your code. The method for assigning and connecting to the Simulator Automation Server can vary depending on the programming environment used, but the idea is basically the same. You define a variable in your program to point to the server object, which is called SimulatorAuto. If the Type Library was imported properly, you should have full access to the SimulatorAuto object and its defined [functions](#). Again, the procedure for creating the object and connecting to SimulatorAuto may vary for different programming languages. Check the help for your programming environment on connecting to COM or Automation servers.

### Examples

The following examples are just a few specific examples for certain programming media. The procedure may be different for other programming media not listed. In addition, a procedure given for a certain type of programming media may be one variation from several possible procedures for accomplishing the same task.

#### Borland Delphi 5

- Add **pwrworld\_TLB** to the **uses** section of your unit.
- Declare a variable globally or as part of another object: **A : ISimulatorAuto.**
- Initialize the variable: **A := nil.**
- To connect to the Simulator Automation Server, create the connection: **A := CoSimulatorAuto.create.**
- Perform function calls to the Simulator Automation Server: **A.SomeFunction(parameters).**
- To close the connection to the Simulator Automation Server, remove the reference by again setting: **A := nil.**

#### Microsoft Visual Basic

- Declare a variable globally or as part of another object or function: **Public A As SimulatorAuto.**
- To connect to the Simulator Automation Server, create the connection: **Set A = New SimulatorAuto.**
- Perform function calls to the Simulator Automation Server: **A.SomeFunction parameters.**
- To close the connection to the Simulator Automation Server, remove the reference: **Set A = Nothing.**

#### Microsoft Visual C++

- Declare a variable globally or as part of another object or function: **IsimulatorAutoPtr \*A.**
- Declare a variable globally or as part of another object or function: **CLSID clsid.**
- Declare a variable globally or as part of another object or function: **HRESULT hr.**
- Obtain the class identifier (clsid) with the following command: **hr = CLSIDFromProgID(L"pwrworld.SimulatorAuto", &clsid).**
- Initialize variable A: **A = new IsimulatorAutoPtr.**
- To connect to the Simulator Automation Server, create the connection: **hr = A>CreateInstance(clsid, NULL, CLSCTX\_SERVER).**
- Perform function calls to the Simulator Automation Server: **A.SomeFunction(parameters).**
- To close the connection to the Simulator Automation Server, release the reference: **hr = A>Release().**



---

### **Simulator Automation Server Properties (version 9)**

---

The following list of parameters is currently available once the SimulatorAuto object is set in your code. Check the help sections on these properties to see more detail on the particular property.

[ExcelApp](#)

---

## ExcelApp Property (version 9)

---

The [Simulator Automation Server](#) has the ability to send data from Simulator to an Excel spreadsheet using the [SendToExcel](#). By default, the Simulator Automation Server starts an instance of Excel the first time one of the above functions is called. Each subsequent call to these two functions will then send data to the same instance of Excel, until it is manually closed by the user. The ExcelApp property allows the user to gain access to the instance of Excel used by the Simulator Automation Server from within their own code. Thus the user can write code to manipulate the external instance of Excel. In addition, the ExcelApp property can be set by the user's code, meaning that the user can initialize an external instance of Excel from within their own code and set the ExcelApp property to their external instance of Excel. Simulator itself is limited to starting only one instance of Excel on its own, but with the ExcelApp property allowing you to set the instance of Excel that Simulator uses on the fly, you can generate multiple instances of Excel within your code, and handle setting the ExcelApp property of the Simulator Automation server to the desired Excel instance depending on the data you want to send to Excel.

The ExcelApp property is a variable of type Variant that returns the pointer to an object representing the external instance of Excel.

### ExcelApp : Variant

To gain access to the external instance of Excel stored in the ExcelApp property, you first need to initialize a variable as an object.

The following are a couple of examples in Borland Delphi and Microsoft Visual Basic.

### Examples

#### Borland Delphi 5

- **Var MyExc : TObject**
- **MyExc = SimAuto.ExcelApp***{Makes the connection to the external instance}*
- *{Perform activities with the Excel instance}*
- **MyExc.Quit***{Closes the external instance if called; do not call if you wish the instance to remain open}*
- **MyExc.Free***{Removes the connection to the external instance}*

#### Microsoft Visual Basic

- **Dim MyExc As Object**
- **Set MyExc = SimAuto.ExcelApp***{Makes the connection to the external instance}*
- *{Perform activities with the Excel instance}*
- **MyExc.Quit***{Closes the external instance if called; do not call if you wish the instance to remain open}*
- **Set MyExc = Nothing***{Removes the connection to the external instance}*

---

## **Simulator Automation Server Functions (version 9)**

---

The following list of functions is currently available once the SimulatorAuto object is set in your code. Check the help sections on these functions to see more detail on the particular function.

[ChangeParameters](#)

[CloseCase](#)

[GetParameters](#)

[ListOfDevices](#)

[LoadContingencies](#)

[OpenCase](#)

[ProcessAuxFile](#)

[RunScriptCommand](#)

[SaveCase](#)

[SendToExcel](#)

[WriteAuxFile](#)

---

## ChangeParameters Function (version 9)

---

The ChangeParameters function allows you to set a list of parameters for a specific object in a case loaded into the [Simulator Automation Server](#). In addition to changing parameters for objects, this function can also be used to set options for some of the Simulator tools, such as [ATC](#) and [OPE](#). This function is identical in setup to the [GetParameters](#) function, with the exception that the Values array must contain a value for each field variable given in the ParamList array.

**ChangeParameters(tObjectType, ParamList, Values, EString)**

### Parameter Definitions

**tObjectType : String** The type of object you are changing parameters for. No default.

**ParamList : Variant** A variant array storing strings. This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables for the specific device, or the device cannot be identified. No Default.

**Values : Variant** A variant array storing variants. This array can store any type of information (integer, string, etc.) in each array position. A value should be passed for each field variable given in the ParamList. The Values array must contain the [key field](#) values for the specific device, or the device cannot be identified. No Default.

**EString : Variant** This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

### Example

**ChangeParameters("gen", [pwBusNum, pwGenID, pwGenAGCable], [1, "1", "Yes"], EString)**

This function call will change the AGC Status of bus number one, generator ID number 1, to "Yes", meaning the generator will be included in AGC if it's area is on AGC control.

---

**CloseCase Function (version 9)**

---

The CloseCase function is used to close a load flow case loaded in the [Simulator Automation Server](#). This function should be called at some point after the [OpenCase](#) function. An error will be returned through the EString parameter if an error occurred while trying to close the case.

**CloseCase(EString)**

**Parameter Definitions**

**EString : Variant** This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.



---

**GetParameters Function (version 9)**


---

The GetParameters function is used to request the values of specified fields for a particular object in the load flow case. For returning field values for multiple objects, you can use a loop to make repeated calls to the GetParameters function, and pass the object and desired field information for each object. This function is identical in setup to the [ChangeParameters](#) function, with the exception that the Values array will be updated with the values for the field variables defined in ParamList.

**GetParameters(tObjectType, ParamList, Values, EString)**

#### Parameter Definitions

**tObjectType : String** The type of object you are changing parameters for. No default.

**ParamList : Variant** A variant array storing strings. This array stores a list of PowerWorld® object field variables, as defined in the section on [PowerWorld Object Fields](#). The ParamList must contain the [key field](#) variables for the specific device, or the device cannot be identified. The remaining field variables in the array define which values to retrieve from Simulator. No Default.

**Values : Variant** A variant array storing variants. This array can store any type of information (integer, string, etc.) in each array position. Values must be passed for the [key field](#) variables in ParamList, in the same array position. The remaining field positions in the Values array can initially be filled with either empty strings or zeroes, as the values will be replaced when the function is processed. No Default.

**EString : Variant** This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

#### Example

**GetParameters("gen", [pwBusNum, pwGenID, pwGenAGCable], ValArray = [1, "1", ""], EString)**

This function call will return the AGC Status of bus number one, generator ID number 1, in the ValArray variable. Note that the generator bus number and ID had to be passed into the function, while the remaining field could be assigned an empty string. If the bus number and ID had not been passed, an error would have been returned since Simulator would not be able to identify a generator. With a valid bus number and ID for the generator, the ValArray variable would have returned an array of the following format: ValArray = [1, "1", "Yes"].

---

**ListOfDevices Function (version 9)**


---

The ListOfDevices function is used to request a list of objects and their [key fields](#) from the [Simulator Automation Server](#). The function can return all devices of a particular type, or can return only a list of devices of a particular type based on an [advanced filter](#) defined for the loaded case. This function is best used in conjunction with a looping procedure and the [ChangeParameters](#) or [GetParameters](#) functions to process a group of devices.

**ListOfDevices(tObjType, EString, filterName, objList1, {objList2}, {objList3})**

### Parameter Definitions

**TobjType : String** The type of object for which you are acquiring the list of devices. No default.

**EString : Variant** This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

**FilterName : String** The name of an [advanced filter](#) defined in the load flow case open in the Simulator Automation Server. If the filter cannot be found, the server will default to returning all objects in the case of type TobjType.

**ObjList1 : Variant** This parameter must be assigned a variable declared as a Variant. The Simulator Simulation Server will return a variant array filled with the first [key field](#) for the devices of type TobjType. This parameter is required for all types of devices.

**ObjList2 : Variant** This parameter **may** be optional. For device types that only have one [key field](#), you can omit passing a variable for this parameter. If the device type requested has two or more key fields, then this parameter must be assigned a variable declared as a Variant. The Simulator Simulation Server will return a variant array filled with the second key field for the devices of type TobjType.

**ObjList3 : Variant** This parameter **may** be optional. For device types that have one or two [key fields](#), you can omit passing a variable for this parameter. If the device type requested has three key fields, then this parameter must be assigned a variable declared as a Variant. The Simulator Simulation Server will return a variant array filled with the third key field for the devices of type TobjType.

### Example

**ListOfDevices("gen", EString, "", objList1, objList2)**

This function call will return a list of all generators in the load flow case. By passing an empty string as the filter name, Simulator will not use an advanced filter before returning the generator key fields. Since generators have two key fields, bus number and ID, it was required to pass two variant array variables to receive the key fields. Each generator in the case would have its bus number returned in objList1 and ID in objList2. The corresponding bus number and ID are stored in the same index position in the two arrays. If only one variant array, objList1, had been passed in this instance, an error would have been returned stating more arrays were necessary. If the third variant array, objList3, had been included, Simulator would have still returned the key fields in objList1 and objList2, and would have returned objList3 as an empty variant.

---

**LoadContingencies Function (version 9)**

---

The LoadContingencies function can be used to read a set of predefined contingencies from a PowerWorld® Auxiliary Contingency file. Thus once you have defined a set of [contingencies](#) in Simulator and saved the list to a PowerWorld® Auxiliary file, you can specify the filename with this function and the [Simulator Automation Server](#) will load the contingencies into memory.

**LoadContingencies(fileName, EString, {tAppend})**

**Parameter Definitions**

**Filename : String** The name of the PowerWorld® Auxiliary file containing the defined contingencies. If the file cannot be found, an error will be returned through EString.

**EString : Variant** This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

**tAppend : Boolean** This parameter is optional. If you have more than one list of contingencies in separate files, you can append the lists together in the Simulator Automation Client by passing the Boolean value True in this parameter. The default if not passed is False, meaning any existing contingencies in the Simulator Automation Client would be cleared prior to loading the new list.

**Example**

**LoadContingencies("c:\my files\ctgfile.aux", EString, True)**

This function call will load a previously created list of contingencies that was saved in the PowerWorld Auxiliary file format. The file name string identifies the location and name of the auxiliary file. By passing the value True in the optional parameter tAppend, Simulator will read the list of contingencies and add them to any existing contingencies already loaded in Simulator. Contingencies in the list existing list with the same name as contingencies in the appending list will be replaced by the records in the appending list.

---

**OpenCase Function (version 9)**

---

The OpenCase function will load a PowerWorld® Simulator load flow file into the [Simulator Automation Server](#). If an error occurs trying to open the case, an error message will be returned through EString.

**OpenCase(filename, EString)**

**Parameter Definitions**

**FileName : String** The name of the PowerWorld® Simulator case file to be loaded into the Simulator Automation Server. This string includes the directory location and full file name. If the file cannot be found or an error occurs while reading the file, an error message will be returned through EString.

**EString : Variant** This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

---

**ProcessAuxFile Function (version 9)**

---

The ProcessAuxFile function will load a PowerWorld® Auxiliary file into the [Simulator Automation Server](#). This allows you to create a text file (conforming to the PowerWorld® Auxiliary file format) that can list a set of data changes and other information for making batch changes in Simulator. If an error occurs while processing the auxiliary file, an error message is returned through EString.

**ProcessAuxFile(filename, EString)**

**Parameter Definitions**

**FileName : String** The name of the PowerWorld® Auxiliary file to be loaded into the Simulator Automation Server. This string includes the directory location and full file name.

**EString : Variant** This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

---

**RunScriptCommand Function (version 9)**

---

The RunScriptCommand function is used to execute a list of [script statements](#). The [script actions](#) are those included in the script sections of the [Auxiliary Files](#). If an error occurs trying to run a script command, an error will be returned through EString.

**RunScriptCommand(Statements, EString)****Parameter Definitions**

**Statements : String** The block of script actions to be executed. Each script statement must end in a semicolon. The block of script actions should **not** be enclosed in curly braces.

**EString : Variant** This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

**Example**

**RunScriptCommand("Entermode(PowerFlow); SolvePowerFlow;", EString)**

This function call will switch to the PowerFlow submode and then will solve the power flow of the present case.

---

## SaveCase Function (version 9)

---

The SaveCase function is used to save a case previously loaded in the [Simulator Automation Server](#) using the [OpenCase](#) function. The function allows you to specify a file name and a format for the save file. If an error occurs while trying to save a case, an error message is returned through EString.

**SaveCase(fileName, EString, {fileType}, {Overwrite})**

### Parameter Definitions

**fileName : String** The name of the file you wish to save as, including file path. No default.

**EString : Variant** This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

**fileType : String** This parameter is optional. If you desire to save the case in a format other than the current PowerWorld Binary (pwb) format, you must specify the type of format in this parameter as a string. If the parameter is not passed, the format type defaults to the most recent version of PowerWorld binary (pwb) file. If a string is passed for this parameter, it must at least be the "PWB" string. An empty string will return an error. The following list is the currently supported list of string identifiers and the file types they represent.

"PTI23" PTI version 23 (raw)  
 "PTI24" PTI version 24 (raw)  
 "PTI25" PTI version 25 (raw)  
 "PTI26" PTI version 26 (raw)  
 "PTI27" PTI version 27 (raw)  
 "GE" GE PSLF (epc)  
 "IEEE" IEEE common format (cf)  
 "PWB70" PowerWorld Binary version 7.0 (pwb)  
 "PWB" PowerWorld Binary (most recent) (pwb)

**Overwrite : Boolean** This parameter is optional. If you do not want to overwrite a file with the same name as passed in filename, you must specify this parameter as False. Default is True.

---

## SendToExcel Function (version 9)

---

The SendToExcel function can be called to send data from the [Simulator Automation Server](#) to an Excel spreadsheet. The function is flexible in that you can specify the type of object data you want to export, an [advanced filter](#) name for a filter you want to use, and as many or as few [field types](#) as desired that are supported by the type of object. The first time this function is called, a new instance of Excel will be started, and the data requested will be pasted to a new sheet. For each subsequent call of this function, the requested data will be pasted to a new sheet within the same workbook, until the workbook is closed. If an error occurs while trying to send data to Excel, an error message is returned through EString.

**SendToExcel(tObjectType, filterName, EString, {tFieldList})**

### Parameter Definitions

**tObjectType : String** A string describing the type of object for which you are requesting data. No Default.

**filterName : String** The name of an [advanced filter](#) which was previously defined in the case before being loaded in the Simulator Automation Server. If no filter is desired, then simply pass an empty string. If a filter name is passed but the filter cannot be found in the loaded case, no filter is used. Default is an empty string.

**EString : Variant** This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

**TFieldList : Variant** This parameter is optional. A variant array of strings, where each string represents an object field variable, as defined in the section on [PowerWorld Object Variables](#). If no array is passed, the Simulator Automation Server will use predefined default fields when exporting the data.

### Example

**SendToExcel("gen", "", EString, [pwBusNum, pwGenID, pwGenAGCable])**

This function call will send the values of the fields in tFieldList to an Excel workbook for all the generators in the load flow case. If a filter name had been passed instead of an empty string, Simulator would have located and used a pre-defined advanced filter and applied it to the information if it was found. By specifying the fields in the optional parameter tFieldList, only the three field values for each generator will be returned. If the optional parameter tFieldList had been omitted, Simulator would have returned internally defined default information for the generators.



---

## WriteAuxFile Function (version 9)

---

The WriteAuxFile function can be used to write data from the case in the [Simulator Automation Server](#) to a PowerWorld® Auxiliary file. The function is flexible in that you can specify the type of object data you want to export, an [advanced filter](#) name for a filter you want to use, and as many or as few field types as desired that are supported by the type of object. In addition, you can specify a new file name for each call to WriteAuxFile, or you can specify the same file name and append the data to the file. If an error occurs while trying to write the auxiliary file, an error message is returned through EString.

```
WriteAuxFile(fileName, filterName, tObjectType, EString, {tAppend},  
{tFieldList})
```

### Parameter Definitions

**FileName : String** The name of the PowerWorld® Auxiliary file you wish to save. No default.

**filterName : String** The name of an [advanced filter](#) which was previously defined in the case before being loaded in the Simulator Automation Server. If no filter is desired, then simply pass an empty string. If a filter name is passed but the filter cannot be found in the loaded case, no filter is used. Default is an empty string.

**tObjectType : String** A string describing the type of object for which you are requesting data. No Default.

**EString : Variant** This parameter must be assigned a variable declared as a Variant. This parameter is for a return value only, and will assign the passed variable a string representing an error that may have occurred in the Simulator Automation Server. If no error occurred, the EString variable will return an empty string.

**TAppend : Boolean** This parameter is optional. If you have given a file name of an auxiliary file that already exists, then the file will either be appended to or overwritten according to the setting of this parameter. If the parameter is not passed, the Simulator Automation Server assumes false.

**TFieldList : Variant** This parameter is optional. A variant array of strings, where each string represents an object field variable, as defined in the section on [PowerWorld Object Variables](#). If no array is passed, the Simulator Automation Server will use predefined default fields when exporting the data.

### Example

```
WriteAuxFile("c:\my files\myauxfile.aux", "", "gen", EString, False,  
[pwBusNum, pwGenID, pwGenAGCable])
```

This function call will send the values of the fields in tFieldList to a PowerWorld Auxiliary file for all the generators in the load flow case. If a filter name had been passed instead of an empty string, Simulator would have located and used a pre-defined advanced filter and applied it to the information if it was found. By specifying the fields in the optional parameter tFieldList, only the three field values for each generator will be returned. If the optional parameter tFieldList had been omitted, Simulator would have returned internally defined default information for the generators. Since the tFieldList optional parameter was included, the tAppend optional parameter also had to be included, even though its default value is already False.

---

## PowerWorld Object Variables (Version 9)

---

The ability to access power system data for different objects through various [Simulator Automation Server functions](#) is based on variables defined in Simulator that can be referred to as Object Field Variables. Each object (i.e. bus, generator, etc.) can have numerous fields associated with it. Each of these fields, in turn, has a variable associated with the field to enable access to the field for the purpose of acquiring or changing data. For example, the [GetParameters](#) function has a parameter called ParamList, which is intended to store a list of Object Field Variables for a particular type of object. When the function is called, the Simulator Automation Server will return the values associated with each particular field variable for the type of object specified. These field variables allow for complete flexibility by the user in specifying as many or as few fields for a particular object when acquiring or changing data.

### Examples of Field Variables

**PWBusGenMW**

**PWBusNum**

Simulator has literally hundreds of parameters spanning numerous types of device and option specifications. Rather than list all of the field variables and the value they represent in this help file, we have enabled Simulator to automatically generate a text file containing the field variables and a description of what value the variable represents. PowerWorld Corporation highly recommends that you examine this list. To generate this text file, run PowerWorld Simulator and access the **Help** menu. Choose the option **Export Object Fields...** Specify a file name and location for saving the file, and click Save. Simulator will save out the field variables, the type of variable (string, integer, etc.), and a description of the value the field variable represents, with key fields for different objects marked with an asterisk. The field variables will also be split into sections based on the type of object they are valid for. Note that the same field variable may be available for more than one object, but that the value represented by the field variable might vary for different objects.

### Examples of Field Variables in Listing

PwBusGenMW Real 'Bus Gen MW'

\* PwBusNum Integer 'Bus Number' (\* denotes key field in text file)

ATC\_MaxLimElements Integer 'Max # Limiting Elements'

The last note on the Object Field Variables is that some of the object field variables are reused for more than one value for an object. For example, a transmission line has "from" and "to" buses associated with the line. Rather than have separate field variables for values at each terminal bus, the same field variable is used for both, with a colon followed by a number appended to the variable to signify which bus the value represents. As an example, consider the field variable for bus per unit voltage, which is PWBusPUVolt. Since there are two buses per line, the "From" bus voltage would be represented as PWBusPUVolt:0, and the "To" bus voltage would be represented as PWBusPUVolt:1. The enumeration of the field variables always starts with 0 for the first instance. You may note that in the text file of field values that you don't see any field variables with :0 appended to them. Since we did not want you to have to always append the :0 on all field variables, the default for a field variable with no appended :# is 0. Thus you would only need to be concerned with appending the :# for field variables that require a number greater than 0.

---

**Chapter 14 : Auxiliary Script/Data Files**

---

Auxiliary Script and Data files are used to automatically change data and control the Simulator in a batch mode fashion.

This chapter covers the following:

- Overview of Auxiliary Files
- Script Command Execution Dialog
- Quick Auxiliary Files Dialog
- Script Section
- Data Section
- Data Section for Display Auxiliary Files

---

## Auxiliary Files

---

PowerWorld has incorporated the ability to import data from data sources other than power flow models into PowerWorld Simulator. Simulator has always had the ability to import supplemental data into a power flow model, but this facility was limited to ten or twelve native data formats targeted to specific applications for which the data had to have a precise format. The extensions described in this help document provide a more versatile interface to data from non-power-flow sources.

In addition to developing auxiliary data formats, a script language was also developed. The script language and auxiliary data formats are incorporated together. This format is described in this help document.

Script/Data files are called *data auxiliary* files in Simulator and typically have the file extension .AUX. These files mostly contain information about power system elements and options for running the various tools within Simulator. They do not contain any information about individual display objects contained on a one-line diagram. There are separate files called *display auxiliary* files that are available for importing/exporting display data from/to Simulator in a text format. These files are distinguished from the *data auxiliary* files by using the extension .AXD. The format for these two types of files is similar, but different object types are supported by each and require that the files be read separately. Currently, there are no script commands that are available for *display auxiliary* files.

Both file types will be generically referred to as auxiliary files. An auxiliary file may be comprised of one or more DATA or SCRIPT sections. A DATA section provides specific data for a specific type of object. A SCRIPT section provides a list of script actions for Simulator to perform.

These sections have the following format:

### **SCRIPT ScriptName1**

```
{
script_statement_1
.
script_statement_n
}
```

### **DATA DataName1(object\_type, [list\_of\_fields], file\_type\_specifier)**

```
{
data_list_1
.
data_list_n
}
```

### **DATA DataName2(object\_type, [list\_of\_fields], file\_type\_specifier)**

```
{
data_list_1
```

```
.  
data_list_n  
}  
  
SCRIPT ScriptName2  
{  
script_statement_1  
.  
script_statement_n  
}
```

Note that the keywords **SCRIPT** or **DATA** must occur at the start of a text file line. Auxiliary files may contain more than one **DATA** or **SCRIPT** section. These sections always begin with the keyword **DATA** or **SCRIPT**. **DATA** sections are followed by an argument list enclosed in ( ). The actual data or script commands are then contained within curly braces { }.

---

## Script Command Execution Dialog

---

The Script Command Execution dialog provides a location for the user to enter [script commands](#) manually, or to load [auxiliary files](#) containing [Script](#) and/or [Data](#) sections previously defined. This dialog is opened by pressing the **Script** button in the **Log** ribbon group on the [Tools](#) ribbon tab.

### Loading Auxiliary Files

The first feature of the Script Command Execution dialog is that it provides a location to load previously defined [auxiliary files](#) for the currently loaded case. The **Auxiliary File** menu provides a location for loading an auxiliary file, or simply validating that the [Script](#) and/or [Data](#) sections of an Auxiliary file are correctly formatted in the file.

The **Quick Aux** option allows you to open the [Quick Auxiliary Files](#) dialog for creating a list of auxiliary files to be opened and processed en masse.

Lastly, you can export the Simulator recognized objects and object fields using the **Export Field Names** option. You can export the field names to a text file or to Excel.

### Running Script Commands

The second feature of the Script Command Execution dialog is that you can run script commands manually. To run a script command, type the command into the display, and press the **Execute** button. Note that if the **Execute on ENTER key** option is checked, the command will also be processed when ENTER is pressed. Note that similar to script command syntax in the [Script](#) section of auxiliary files, script commands must be ended with a semi-colon (;) in the Script Command Execution dialog as well.

You can enter multiple script commands to be processed in sequence in this display. To do so, you must uncheck the option **Execute on ENTER key**. Then you can press enter after each script command to move to the next line and enter another command. Use the **Execute** button to process the sequence of script commands.

If you are running a sequence of script commands and wish to abort the run, use the **Abort** button.

If you wish to view the message log while script commands are processing, open the log using the **Show Log** button.

---

## Quick Auxiliary Files Dialog

---

The Quick Auxiliary Files dialog can be accessed from the **Quick Aux Files** menu option of the [Script Command Execution](#) dialog. This dialog gives you a location for creating a list of auxiliary files to be processed en masse for the currently loaded case.

Using the **Define** option of the **Quick Aux Files** menu, the Quick Auxiliary Files dialog will open. Use this dialog to **Add** previously defined [auxiliary files](#) to the list of files to be processed. You can rearrange the order of the files by selecting a file and using the up and down arrows on the right to move the selected file within the list, or you can sort them alphabetically using the **Sort** button. To remove an auxiliary file from the list, use the **Delete** button.

Once the list of auxiliary files to process is complete, press the **Execute** button to process the list of auxiliary files.

---

**Script Section**


---

The SCRIPT section begins with a left curly brace and ends with a right curly brace.

**SCRIPT ScriptName**

```
{
script_statement_1
script_statement_2
.
.
.
script_statement_n
}
```

Scripts may optionally contain a ScriptName. This enables you to call a particular SCRIPT by using the [LoadScript action](#). After the optional name, the SCRIPT section begins with a left curly brace and ends with a right curly brace. Inside of this, script statements can be given.

In general, a script statement has the following format:

**Keyword(arg1, arg2, ...);**

- Statement starts with a keyword.
- The keyword is followed by an argument list which is encompassed in parentheses ( ).
- The arguments are separated by commas.
- If a single argument is a list of things, this list is encompassed by braces [ ]. (eg. SetData)
- Statements end with a semicolon.
- Statements may take up several lines of the text file.
- You may put more than one statement on a single text line.

Some features in Simulator are available exclusively in either [RUN](#) mode or [EDIT](#) mode. This functionality is preserved in the script language, but with the addition of a **submode** feature. Submodes limit what script commands can be called. Only those commands available to the current submode can be executed. You will always be in one of the submodes when executing a script. If the Script Command Execution Dialog is opened from Edit Mode, Simulator defaults to the **EDIT, CASE** submode. If the Script Command Execution Dialog is opened from Run Mode (or when a script is initially started), Simulator defaults to the **RUN, POWERFLOW** submode.

To switch submodes, use the [EnterMode \(mode or submode\)](#) script command. Available submodes include:

Edit Mode

Case Submode

Run Mode

PowerFlow Submode

Contingency Submode

ATC Submode

Fault Submode



PV Submode  
QV Submode

---

**Script General Actions**


---

The following actions are available to you regardless of the Mode or SubMode:

RenameFile ("oldfilename", "newfilename");

CopyFile ("oldfilename", "newfilename");

DeleteFile ("filename");

LoadAux ("filename", CreateIfNotFound);

LoadScript ("filename", ScriptName);

LoadData ("filename", DataName, CreateIfNotFound);

SelectAll (objecttype, filter);

UnSelectAll (objecttype, filter);

Delete (objecttype, filter);

DeleteIncludingContents (objecttype, filter);

SaveData("filename", filetype, objecttype, [fieldlist], [subdatalist], filter, [SortFieldList]);

SaveDataWithExtra("filename", filetype, objecttype, [fieldlist], [subdatalist], filter, [SortFieldList], [Header\_List], [Header\_Value]);

SaveDataUsingExportFormat("filename", filetype, FormatName);

SetData (objecttype, [fieldlist], [valuelist], filter);

CreateData (objecttype, [fieldlist], [valuelist]);

ChangeData (objecttype, [fieldlist], [valuelist], filter); (NOT AVAILABLE YET)

WriteTextToFile ("filename", "text...");

SetCurrentDirectory ("filedirectory", CreateIfNotFound);

ExitProgram;

NewCase;

OpenCase ("filename"); // assumes to open as PWB

OpenCase	("filename",	openfiletype);		
		PWB, GE, PTI, or CF		
SaveCase	("filename");	// assumes to save as PWB		
SaveCase	("filename",	savefiletype);		
		PWB, PWB5, PWB6, PWB7,PWB8, PWB9, PWB10, PWB11, PWB12		
		PTI23, PTI24, PTI25, PTI26, PTI27,PTI28, PTI29, PTI30		
		GE, GE14, GE15		
		CF		
EnterMode	(mode or	submode);		
	EDIT	CASE		
	RUN	POWERFLOW		
		CONTINGENCY		
		ATC		
		FAULT		
		PV		
		QV		
OpenOneLine	("filename",	"view",	FullScreen);	
ExportOnlineAsShapeFile	("filename",	"OnlineName",	"ShapeFileExportDescriptionName",	Use YES
LogClear;				
LogSave	("filename",	AppendFile);		
		YES or NO		
LogAdd	("string...");			
LogAddDateTime	("label",	IncludeDate,	IncludeTime,	IncludeMilliseconds);
		YES or NO	YES or NO	YES or NO

CaseDecsriptioClear;

CaseDescriptionSet ("text...", Append  
YES or NO

SaveYbusInMatlabFormat ("filename", IncludeVoltages);  
YES or NO

SaveJacobian ("Jacfilename", "JIDfilename", filetype, JacForm);  
M R  
TXT P

SetParticipationFactors (Method, ConstantValue, Object);  
MAXMWRAT [Area num]  
RESERVE [Zone num]  
CONSTANT value SYSTEM  
AREAZONE or  
DISPLAYFILTERS

GenForceLDC\_RCC (filter);

CalculateRXBGFromLengthConfigCondType (filter);

DirectionsAutoInsert (Source, Sink, DeleteExisting, UseDisplayFilters,  
AREA AREA YES or NO YES or NO  
ZONE ZONE  
INJECTIONGROUP INJECTIONGROUP  
SLACK

DeterminePathDistance ([start], BranchDistMeas, BranchFilter, BusField);

DetermineShortestPath ([start], [end], BranchDistMeas, BranchFilter, FileName);

**RenameFile("oldfilename", "newfilename");**

Use this action to rename a file from within a script.

**"oldfilename"**: The present file name.

**"newfilename"**: The new file name desired.

**CopyFile("oldfilename", "newfilename");**

Use this action to copy a file from within a script.

**"oldfilename"**: The present file name.

**"newfilename"**: The new file name desired.

**DeleteFile("filename");**

Use this action to delete a file from within a script.

**"filename"**: The file name to delete.

**LoadAux("filename", CreateIfNotFound);**

Use this action to load another auxiliary file from within a script.

**"filename"**: The filename of the auxiliary file being loaded.

**CreateIfNotFound**: Set to YES or NO. YES means that objects that cannot be found will be created while reading in DATA sections from filename. If this parameter is not specified, then NO is assumed.

**LoadScript("filename", ScriptName);**

Use this action to load a named Script Section from another auxiliary file. This will open the auxiliary file denoted by "filename", but will only execute the script section specified.

**"filename"**: The filename of the auxiliary file being loaded.

**ScriptName**: The specific ScriptName from the auxiliary file which should be loaded.

**LoadData("filename", DataName, CreateIfNotFound);**

Use this action to load a named Script Section from another auxiliary file. This will open the auxiliary file denoted by "filename", but will only execute the script section specified.

**"filename"**: The filename of the auxiliary file being loaded.

**DataName**: The specific ScriptName from the auxiliary file which should be loaded.

**CreateIfNotFound**: Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections from filename. If this parameter is not specified, then NO is assumed.

**SelectAll(objecttype, filter);**

Use this to set the selected property of objects of a particular type to true. A filter may optionally be specified to only set this property for objects which meet a filter.

**objecttype**: The objecttype being selected.

**filter**: There are three options for the filter:

**SelectAll(objecttype);**: No filter specified means to select all objects of this type.

**SelectAll(objecttype, "filtername");**: "filtername" means select those that meet the filter.

**SelectAll(objecttype, AREAZONE);** AREAZONE means select those that meet the area/zone filters.

**UnSelectAll(objecttype, filter);**

Same as SelectAll, but this action sets the *selected* properties to false.

**Delete(objecttype, filter);**

Use this to delete objects of a particular type. A filter may optionally be specified to only delete objects that meet a filter.

**objecttype:** The objecttype being selected.

**filter:** There are four options for the filter:

**Delete(objecttype);** No filter specified means to delete all objects of this type.

**Delete(objecttype, "filtername");** "filtername" means delete those that meet the filter.

**Delete(objecttype, AREAZONE);** AREAZONE means delete those that meet the area/zone filters.

**Delete(objecttype, SELECTED);** SELECTED means delete those objects whose *selected* field is set to YES.

**DeleteIncludingContents(objecttype, filter);**

Use this to delete objects of a particular type and other objects that these contain. Currently, only multi-section lines (objecttype = MultiSectionLine) can be used with this command. The branches and dummy buses that belong to multi-section lines will also be deleted along with the multi-section lines. A filter may optionally be specified to only delete objects that meet a filter.

**objecttype:** The objecttype being selected (presently we only support objecttype = MultiSectionLine).

**filter:** There are four options for the filter:

**DeleteIncludingContents(objecttype);** No filter specified means to delete all objects of this type.

**DeleteIncludingContents(objecttype, "filtername");** "filtername" means delete those that meet the filter.

**DeleteIncludingContents(objecttype, AREAZONE);** AREAZONE means delete those that meet the area/zone filters.

**DeleteIncludingContents(objecttype, SELECTED);** SELECTED means delete those objects whose *selected* field is set to YES.

**SaveData("filename", filetype, objecttype, [fieldlist], [subdatalist], filter, [SortFieldList]);**

Use this action to save data in a custom defined format. A filter may optionally be specified to save only objects which meet a filter, and a sort order list may also optionally be specified that will allow saving the objects in a particular order.

**"filename":** The file to save the data to

**filetype:**

AUX (or AUXCSV): save as a space-delimited (or comma-delimited) auxiliary data file.  
 CSV: save as a normal CSV text file, without the AUX file formatting. The first few lines of the text file will represent the object name and field names.

**objecttype:** The objecttype being saved.

**[fieldlist]:** A list of fields to save. For numeric fields, the number of digits and the number of decimal places (digits to right of decimal) can be specified by using the following format for the field, `variablename:location:digits:rod`.

**[subdatalist]:** A list of the subdata objecttypes to save with each object record.

**filter:** There are four options for the filter:

**SaveData(...);:** No filter specified means to save all objects of this type.

**SaveData(..., "filtername");:** "filtername" means save those that meet the filter.

**SaveData(..., AREAZONE);:** AREAZONE means save those that meet the area/zone filters.

**SaveData(..., SELECTED);:** SELECTED means save those objects whose *selected* field is set to YES.

**[SortFieldList]:** This allows the specification of a sort order in which the data will be saved. The format is:

**[variablename1:+:0, variablename2:-:1]**

where

**variablename** is the name of the field to sort by. There is no limit to how many fields can be specified for sorting. For fields that require a location other than zero, **variablename** can be in the format **fieldname:location**.

+ or - for the second parameter indicates sort ascending for + and sort descending for -. This parameter must be specified.

0 or 1 for the third parameter indicates: 0 - case insensitive and do not use absolute value, 1 - case sensitive or use absolute value. This parameter is optional.

**SaveDataWithExtra("filename", filetype, objecttype, [fieldlist], [subdatalist], filter, [SortFieldList], [Header\_List], [Header\_Value\_List]);**

Use this action to save data in a custom defined format. User-specified fields and field values can also be specified in the output. Optional parameters are filter, SortFieldList, Header\_List, and Header\_Value\_List.

**"filename":** The file to save the data to

**filetype:**

CSV: save as a normal CSV text file, without the AUX file formatting. The first few lines of the text file will represent the object name and field names.

CSVNOHEADER: save as a normal CSV text file, without the AUX file formatting. The object name and field names are NOT included. This option is useful when appending data of the same object type and field list into a common file.

Data cannot be saved using the AUX filetype with this command.

**objecttype:** The objecttype being saved.

**[fieldlist]:** A list of fields to save. For numeric fields, the number of digits and the number of decimal places (digits to right of decimal) can be specified by using the following format for the field, `variablename:location:digits:rod`.

**[subdatalist]:** A list of the subdata objecttypes to save with each object record.

**filter:** There are four options for the filter:

**SaveData(...);** No filter specified means to save all objects of this type.

**SaveData(..., "filtername");** "filtername" means save those that meet the filter.

**SaveData(..., AREAZONE);** AREAZONE means save those that meet the area/zone filters.

**SaveData(..., SELECTED);** SELECTED means save those objects whose *selected* field is set to YES.

**[SortFieldList]:** This allows the specification of a sort order in which the data will be saved. The format is:

**[variablename1:+:0, variablename2:-:1]**

where

**variablename** is the name of the field to sort by. There is no limit to how many fields can be specified for sorting. For fields that require a location other than zero, **variablename** can be in the format **fieldname:location**.

**+** or **-** for the second parameter indicates sort ascending for **+** and sort descending for **-**. This parameter must be specified.

**0** or **1** for the third parameter indicates: **0** - case insensitive and do not use absolute value, **1** - case sensitive or use absolute value. This parameter is optional. If it is not specified, the default is 0.

**[Header\_List]:** This allows the specification of user-defined fields that will appear in the output. Headers should be specified as a list of comma delimited strings. A string should be enclosed in double quotes if the string contains a comma. Header strings cannot be blank.

**[Header\_Value\_List]:** Allows the specification of the values that should be assigned to the user-defined fields specified by Header\_List. Specifying the values is optional. If specified, there must be as many values specified as there are headers. If not specified, all values are blank. Each object will use the same specified value for the specified field. To use different values for different objects and save these in the same file, make use of the CSVNOHEADER file format and filtering.

**SaveDataUsingExportFormat("filename", filetype, FormatName);**

Use this action to save data in a user-defined format that has previously been defined.

**"filename":** The file to save the data to

**filetype:**

AUX (or AUXCSV): save as a space-delimited (or comma-delimited) auxiliary data file.

CSV: save as a normal CSV text file, without the AUX file formatting. The first few lines of the text file will represent the object name and field names.

**FormatName:** The name of the Object Export Format Description to use.

**SetData(objecttype, [fieldlist], [valuelist], filter);**

Use this action to set fields for particular objects. If a filter is specified, then it will set the respective fields for all objects which meet this filter. Otherwise, if no filter is specified, then the keyfields must be included in the field list so that the object can be found.

**objecttype:** The objecttype being set.

**[fieldlist]:** A list of fields that you want to save.



**[valuelist]:** A list of values to set the respective fields to.

**filter:** There are four options for the filter:

**SetData(...);:** No filter specified: set data only for the object described by the [fieldlist] and [valuelist] parameters.

**SetData(..., "filtername");:** "filtername": set data for all objects that meet the filter.

**SetData(..., AREAZONE);:** AREAZONE: set data for all objects that meet the area/zone filters.

**SetData(..., SELECTED);:** SELECTED: set data for all objects whose *selected* field is set to YES.

**CreateData(objecttype, [fieldlist], [valuelist]);**

Use this action to create particular objects. Note that the key fields for the objecttype must be specified.

**objecttype:** The objecttype being set.

**[fieldlist]:** A list of fields that you want to save.

**[valuelist]:** A list of values to set the respective fields to.

**WriteTextToFile("filename", "text...");**

Use this action to write text to a file. If the specified file already exists, the text will be appended to the file. Otherwise, it creates the file and writes the text to the file.

**"filename":** The file path and name to save.

**"text...":** The text to be written to the file.

**SetCurrentDirectory("filedirectory", CreateIfNotFound);**

Use this action to set the current working directory.

**"filedirectory":** The path of the work directory.

**CreateIfNotFound:** Set to YES or NO. YES means that if the directory path can not be found, the directory will be created. If this parameter is not specified, then NO is assumed.

**ExitProgram;**

Immediately exits the program with no prompts.

**NewCase;**

This action clear out the existing case and open a new case from scratch.

**OpenCase("filename", OpenFileType);**

This action will open a case stored in "filename" of the type OpenFileType;

**"filename":** The file to be opened.

**OpenFileType:** An optional parameter saying the format of the file begin opened is. If none is specified, then PWB will be assumed. It may be one of the following strings:

PWB, PTI, GE, CF, AUX.

**SaveCase("filename", SaveFileType);**

This action will save the case to "filename" in the format SaveFileType.

**"filename"**: The file name to save the information to.

**SaveFileType**: An optional parameter saying the format of the file to be saved. If none is specified, then PWB will be assumed. It may be one of the following strings:

PWB, PWB5, PWB6, PWB7, PWB8, PWB9, PWB10, PWB11, PWB12

PTI (means PTI30), PTI23, PTI24, PTI25, PTI26, PTI27, PTI28, PTI29, PTI30

GE (means GE15), GE14, GE15

CF

AUX

**EnterMode(mode or submode);**

This action will tell Simulator what mode or submode to enter before performing the next script command.

**SubMode**: The submode to enter. A parameter stating what submode to put the program in. Options available are CASE, POWERFLOW, CONTINGENCY, ATC, FAULT, PV, QV. One may also put in RUN or EDIT which will place the program in the POWERFLOW or CASE respectively.

**OpenOnline("filename", "view", FullScreen);**

Use this action to open a online diagram.

**"filename"**: The file name of the online diagram to open.

**"view"**: The view name that should be opened. Pass an empty string to denote no specific view.

**FullScreen**: Set to YES or NO. YES means that the online diagram will be opened in full screen mode. If this parameter is not specified, then NO is assumed.

**ExportOnlineAsShapeFile("filename", "OnlineName",  
"ShapeFileExportDescriptionName", UseLonLat);**

Use this action to save an open online diagram to a shapefile.

**"filename"**: The file name of the shapefile to save.

**"OnlineName"**: The name of the online diagram to save to a shapefile. The online diagram must be open. Use the OpenOnline script command if necessary to open the appropriate online.

**"ShapeFileExportDescriptionName"**: Name of the ShapeFile Export Description to use when saving the shapefile.

**UseLonLat**: Set to YES or NO. YES means that the coordinates of objects on the online diagram will be saved using longitude,latitude. This will only be true if a valid map projection is in use with the online diagram. Otherwise, the coordinates will be saved in x,y. If this parameter is set to NO, the coordinates will be saved in x,y. If this parameter is not specified, YES is assumed.

**LogClear;**

Use this action to clear the Message Log.

**LogSave("filename", AppendFile);**

This action saves the contents of the Message Log to "filename".

**"filename"**: The file name to save the information to.

**AppendFile**: Set to YES or NO. YES means that the contents of the log will be appended to "filename". NO means that "filename" will be overwritten.

**LogAdd("string...");**

Use this action to add a personal message to the MessageLog.

**"string..."**: The string that will appear as a message in the log.

**LogAddDateTime("label", IncludeDate, IncludeTime, IncludeMilliseconds);**

Use this action to add a personal message to the MessageLog, including the posting date/time.

**"label"**: A string that will appear at the start of the line containing the date/time.

**IncludeDate**: Set to YES or NO. YES means that the date will be added after the string specified. If this parameter is not specified, then YES is assumed.

**IncludeTime**: Set to YES or NO. YES means that the time will be added after the date. If there's no date, then it will be added after the string specified. If this parameter is not specified, then YES is assumed.

**IncludeMilliseconds**: Set to YES or NO. YES means that the time including milliseconds will be added after the date or string specified. This is valid only if the IncludeTime is set to YES. If this parameter is not specified, then YES is assumed.

**CaseDescriptionClear;**

Use this action to clear the case description of the presently open case.

**CaseDescriptionSet("text...", Append);**

Use this action to set or append text to the case description

**"text..."**: Specify the text to set/append to the case description

**Append**: YES will append the text to the existing case description. NO will replace the case description

**SaveYbusInMatlabFormat("filename", IncludeVoltages);**

Use this action to store the power system bus admittance matrix (Ybus) in a text format that can be easily read into other programs such as MATLAB®.

**"filename"**: The file name to save the information to. The Ybus data is stored using the MATLAB sparse matrix format in the matrix Ybus.

**IncludeVoltages**: Set to YES or NO. YES means that the bus voltages are also stored but in the vector V.

**SaveJacobian("filename", IncludeVoltages);**

Use this action to store the power flow Jacobian in a text format that can be easily read into other programs such as MATLAB®.

**"Jacfilename":** The file name to save the Jacobian data to. The Jacobian is stored using the MATLAB sparse matrix format in the matrix Jac.

**"JIDfilename":** The file name to save the text identifier information to. This information is used to translate the bus numbering convention used in the Jacobian and Ybus files with the actual bus number and name in the case.

**filetype:** A parameter saying the format of the file to be saved. It may be one of the following strings:

M: MATLAB .M Format.

TXT: ASCII format.

**JacForm:** Specifies the form of the Jacobian. It may be one of the following strings:

R: Rectangular form

P: Polar form.

### **SetParticipationFactors(Method, ConstantValue, Object);**

Use this action to define the participation factors for multiple generators.

**Method:** The formula used to calculate the participation factors for each generator. It may be one of the following strings:

MAXMWRAT: The participation factor for each generator is set to the generator's maximum MW capability.

RESERVE: The participation factor for each generator is set to the generator's reserve power, so that each generator participates in proportion to how much it has left to contribute (Max MW rating - Present MW).

CONSTANT: The participation factor for each generator is set to the same specified value.

**ConstantValue:** The value used if CONSTANT method is specified. If CONSTANT method is not specified, enter 0 (zero).

**Object:** The set of generators to which the participation factors will be assigned. It may be one of the following settings:

[Area num], [Area "name"] [Area "label"]: For all the generators in area

[Zone num], [Zone "name"] [Zone "label"]: For all the generators in zone

SYSTEM: For all the generators in the system regardless of their area or zone affiliation.

AREAZONE or DISPLAYFILTERS: For just the generators whose area/zone/owner filter criteria evaluates to true.

### **GenForceLDC\_RCC(filter);**

Use this action to convert the voltage setpoint for Line Drop and Reactive Current Compensation (LDC/RCC) for multiple generators.

**filter:** There are four options for the filter:

**GenForceLDC\_RCC;:** No filter specified means to convert voltage setpoint for LDCC/RCC on all the generators of the system.

**GenForceLDC\_RCC ("filtername");**: "filtername" means to convert voltage setpoint for LDCC/RCC on those generators that meet the filter.

**GenForceLDC\_RCC (AREAZONE);**: AREAZONE means to convert voltage setpoint for LDCC/RCC on those generators that meet the area/zone filters.

**GenForceLDC\_RCC (SELECTED);**: SELECTED means to convert voltage setpoint for LDCC/RCC on those generators whose *selected* field is set to YES.

#### **CalculateRXBGFromLengthConfigCondType(filter);**

Use this action to go through branches in the power system and automatically recalculate the per unit R, X, G, and B values using the TransLineCalc tool. The branches Conductor Type, Tower Configuration, and Line Length will be passed to the TransLineCalc tool and new R, X, G and B values will be calculated. This is only available if you have installed the TransLineCalc tool.

**filter:** There are four options for the filter which operate identically to the option for the GenForceLDC\_RCC action above.

**CalculateRXBGFromLengthConfigCondType;** No filter specified means to perform calculation on all branches.

**CalculateRXBGFromLengthConfigCondType ("filtername");**: "filtername" means to perform calculation on those branches that meet the filter.

**CalculateRXBGFromLengthConfigCondType;** AREAZONE means to perform calculation on those branches that meet the area/zone filters.

**CalculateRXBGFromLengthConfigCondType:** SELECTED means to perform calculation on those branches whose *selected* field is set to YES.

#### **DirectionsAutoInsert(Source, Sink, DeleteExisting, UseDisplayFilters, Start, Increment);**

Use this action to automatically insert multiple directions for PTDF studies.

**Source, Sink:** The type of object used as the source/sink of the directions. Currently, only the following types of directions can be automatically defined:

AREA SLACK: Directions from areas to slack bus of the system.

ZONE SLACK: Directions from zones to slack bus of the system.

INJECTIONGROUP SLACK: Directions from injection groups to slack bus of the system.

AREA AREA: Directions from areas to other areas of the system.

ZONE ZONE: Directions from zones to other zones of the system.

INJECTIONGROUP INJECTIONGROUP: Directions from injection groups to other injection groups of the system.

**DeleteExisting:** Set to YES or NO. YES means that previously defined directions will be deleted before the new directions are automatically inserted. NO means automatically inserted directions will be added to the list of previously defined directions. If this parameter is not specified, then YES is assumed.

**UseDisplayFilters:** Set to YES or NO. YES means that only Areas and Zones with their Area/Zone filters set to YES will be used when automatically inserting directions. NO specifies all Areas and Zones will be used to automatically insert directions regardless of their Area/Zone filters. If this parameter is not specified, then NO is assumed.

**Start:** Integer value that specifies the first number used when automatically numbering the automatically inserted directions. If this parameter is not specified, then 1 is assumed.

**Increment:** Integer value that will be used as increment in numbering the subsequent automatically inserted directions. If this parameter is not specified, then 1 is assumed.

#### **DeterminePathDistance([start], BranchDistMeas, BranchFilter, BusField);**

Use this action to calculate a distance measure at each bus in the entire model. The distance measure will represent how far each bus is from the starting group specified. The distance measure can be related to impedance, geographical distance, or simply the number of nodes.

**[start]** : is an the starting place which is enclosed in brackets. The starting place may be either an Area, Zone, SuperArea, Substation, Injection Group, or Bus.

**BranchDistMeas** : is either X, Z, Length, Nodes, or a variable name for a branch.

**X** : means use the series reactance,

**Z** : means use  $\sqrt{X^2 + R^2}$ ,

**Length** : means us the Length field, and

**Nodes** : means treat each branch as a length of one.

**variablename** : Otherwise use any Branch object variable name.

**BranchFilter** : is either All, Selected, Closed or the name of a branch Advanced Filter. This parameter is used to specify which branch can be traversed at all.

**All** : means all branches can be traversed

**Selected** : means only branch whose Selected field is YES can be traversed

**Closed** : means only branches that are CLOSED can be traversed.

**Filtername** : means only branches who meet the Advanced Filter named can be traversed.

**BusField** : is the variablename of a Bus field. This field is populated with the minimum distance from the Start Place to that bus. All buses in the start group will have a distance measure of zero. Buses which can not be reached from the start group will have a distance measure of -1.

#### **DetermineShortestPath([start], [end], BranchDistanceMeasure, BranchFilter, Filename);**

Use this action to calculate the shortest path between a starting group and an ending group. The results will be written to a textfile specified by filename. In the text file, the first bus listed will be in the end grouping and the last bus listed will be the start grouping. The result text file will have a line for each bus passed. Each line will contain three entries delimited by a space: "Number DistanceMeasure Name".

**[start]** : same as the starting place for the DeterminePathDistance script command

**[end]** : same as the starting place for the DeterminePathDistance script command

**BranchDistanceMeasure** : same as for DeterminePathDistance script command

**BranchFilter** : same as for DeterminePathDistance script command

**Filename** : is a filename (may need to be enclosed in quotes) to which the results will be written.

---

## Script Edit Mode Actions

---

The following script commands are available during the Case submode of Edit mode:

Equivalence;

DeleteExternalSystem;

SaveExternalSystem ("filename", Savefiletype, withties);

Scale (scaletype, basedon, [parameters], ScaleMarker);

LOAD MW [P,Q] BUS

GEN FACTOR [P] (means constant pf) AREA

INJECTIONGROUP ZONE

BUSSHUNT [P, +Q, -Q]

Move ([elementA], [destination parameter]);

[GEN numA idA] [numB idB]

[LOAD numA idA] [numB idB] // NOT AVAILABLE YET

[BRANCH numA1 numA2 cktA] [numB1 numB2 cktB] // NOT AVAILABLE YET

Combine ([elementA], [elementB]);

[GEN numA idA] [GEN numB idB]

[LOAD numA idA] [LOAD numB idB] // NOT AVAILABLE YET

[BRANCH numA1 numA2 cktA] [BRANCH numB1 numB2 cktB] // NOT AVAILABLE YET

SplitBus ([element], NewBusNumber, InsertBusTieLine, LineOpen);

[BUS num] num YES or NO YES or NO

MergeBuses (NewBusNumber, filter);

	num				
TapTransmissionLine	([element], [BRANCH numA1 numA2 cktA];	PosAlongTheLine,  Value in %	NewBusNumber,  num	ShuntModel,  LINESHUNTS	TreatA  YES or
				CAPACITANCE	
InterfacesAutoInsert	(Type,  AREA  ZONE	DeleteExisting,  YES or NO	UseDisplayFilters,  YES or NO	Prefix,  "string"	Limits) ;  AUTO  ZEROS  [value 1, ... value 8]

## Equivalence

This action will equivalence a power system. All options regarding equivalencing are handled by the Equiv\_Options objecttype. Use the SetData() action, or a DATA section to set these options prior to using the Equivalence() action. Also, remember that the property BusEquiv must be set true for each bus that you want to equivalence.

## DeleteExternalSystem

This action will delete part of the power system. It will delete those buses whose property BusEquiv must is set true.

## SaveExternalSystem("filename", SaveFileType, WithTies)

This action will save part of the power system to a "filename". It will save only those buses whose property BusEquiv must is set true.

**"filename"**: The file name to save the information to.

**SaveFileType**: An optional parameter saying the format of the file to be saved. If none is specified, then PWB will be assumed. My be one of the following strings:

PWB, PWB5, PWB6, PWB7. PWB8, PWB9, PWB10, PWB11, PWB12

PTI (means PTI30) PTI23, PTI24, PTI25, PTI26, PTI27, PTI28, PTI29, PTI30

GE (means GE15), GE14, GE15

CF



## AUX

**WithTies:** An optional parameter. One must specify the file type explicitly in order to use the WithTies parameter. Allows one to save transmission lines that tie the a bus marked with BusEquiv as false and one marked true. This must be a string which starts with the letter Y, otherwise NO will be assumed.

**Scale(scaletype, basedon, [parameters], scalemarker);**

Use this action to scale the load and generation in the system.

**scaletype:** The objecttype begin scaled. Must be either LOAD, GEN, INJECTIONGROUP, or BUSSHUNT.

**basedon:** MW: parameters are given in MW, MVAR units.

FACTOR: parameters a factor to multiple the present values by.

**[parameters]:** These parameters have different meanings depending on ScaleType.

LOAD: [MW, MVAR] or [MW]. If you want to scale load using constant power factor, then do not specifying a MVAR value.

GEN: [MW].

INJECTIONGROUP: [MW, MVAR] or [MW] . If you want to scale load using constant power factor, then do not specifying a MVAR value.

BUSSHUNT: [GMW, BCAPMVAR, BREAMVAR]. The first values scales G shunt values, the second value scales positive (capacitive) B shunt values, and the third value scales negative (reactive) B shunt values.

**scalemarker:** This value specifies whether to look at an element's bus, area, or zone to determine whether it should be scaled.

BUS: Means that elements will be scaled according to the BusScale property of the element's terminal bus.

AREA: Means that elements will be scaled according to the BGScale property of the element's Area. Note that it is possible for the area of a load, generator, or switched shunt to be different than the terminal bus's area.

ZONE: Means that elements will be scaled according to the BGScale property of the element's Zone. Note that it is possible for the zone of a load, generator, or switched shunt to be different than the terminal bus's zone.

OWNER: Means that the elements will be scaled according to the BGScale property of the element's Owner.

**Move([elementA], [destination parameters]);**

NOTE: THIS ACTION IS ONLY AVAILABLE FOR GENERATORS.

Use this action to move a generator, load, or transmission line.

**[elementA]:** the object that should be moved. Must be one of the following formats.

[GEN busnum id], [GEN "name\_nomkv" id], [GEN "buslabel" id], [GEN "label"]

[LOAD busnum id], [LOAD "name\_nomkv" id], [LOAD "buslabel" id], [LOAD "label"]

[BRANCH busnum1 busnum2 ckt], [BRANCH "name\_kv1" "name\_kv2" ckt],

[BRANCH "buslabel1" "buslabel2" ckt], [BRANCH "label"]

**[destination parameters]:** These parameters have different meanings depending on object type of the element. Bus numbers must be used here.

GEN: [busnum id]  
LOAD: [busnum id]  
BRANCH: [busnum1 busnum2 id]

**Combine([elementA], [elementB]);**

NOTE: THIS ACTION IS ONLY AVAILABLE FOR GENERATORS.

Use this action to combine two generators, two loads, or two transmission line. Note that elementA and elementB must be of the same object type. You can not combine a BRANCH and a LOAD.

**[elementA]:** The object that should be moved. Must be one of the following formats.

[GEN busnum id]  
[LOAD busnum id]  
[BRANCH busnum1 busnum2 ckt]

**[elementB]:** The object that element A should be combined with. Must the same format as for elementA.

**SplitBus([element], NewBusNumber, InsertBusTieLine, LineOpen);**

Use this action to transform one bus into two connected buses.

**[element]:** The bus that should be split. Must be of the following format:

[BUS num]  
[BUS "name\_nomkv"]  
[BUS "buslabel"]

**NewBusNumber:** The number of the new bus resulting from the split. Must be unique, that is, it cannot be a number that identifies another number in the case.

**InsertBusTieLine** Set to YES or NO. YES means to insert a very low-impedance bus tie between the bus to split and its offspring. The new branch will have an impedance of  $0.0000 + j0.0001$  ohms. NO means that no bus tie will be inserted. If this parameter is not specified, then YES is assumed.

**LineOpen** Set to YES or NO. YES means that the bus tie should be inserted as an open branch. NO means that the bus tie will be inserted as a closed branch. This options is valid only if InsertBusTieLine is set to YES. If this parameter is not specified, then NO is assumed.

**MergeBuses([element], filter);**

Use this action to merge a set of buses into a single bus.

**[element]:** The number of the new bus resulting from the merging. It should be a number of a bus that is part of the set of buses to be merged, or it should be unique, meaning that it cannot be a number that identifies another number in the case. Must be of the following format:

[BUS num]

**filter:** There are four options for the filter:

**MergeBuses(...);** No filter specified means to merge all the buses of the system.

**MergeBuses(..., "filtername");** "filtername" means to merge those buses that meet the filter.

**MergeBuses(..., AREAZONE);** AREAZONE means to merge those buses that meet the area/zone filters.

**MergeBuses(..., SELECTED);** SELECTED means to merge those buses whose *selected* field is set to YES.

**TapTransmissionLine([element], PosAlongTheLine, NewBusNumber, ShuntModel, TreatAsMSLine);**

Use this action to insert a bus at some location along an existing transmission line.

**[element]:** The transmission line to tap. Must be of the following format:

[BRANCH busnum1 busnum2 ckt]

[BRANCH "name\_kv1" "name\_kv2" ckt]

[BRANCH "buslabel1" "buslabel2" ckt]

[BRANCH "label"]

**PosAlongTheLine:** The value that indicates the point where the new bus is to be inserted relative to the location of the nearest of the two buses in terms of a percentage of the total line length. This value controls how the impedances of the new lines are set, as the impedance of each section will equal the section's corresponding percentage length multiplies by the impedance of the original line.

**NewBusNumber:** The number of the new bus resulting from the tapping. Must be unique, that is, it cannot be a number that identifies another number in the case.

**ShuntModel:** The way the original charging capacitance of the line is reassigned. The choices are:

**LINEHUNTS:** The original charging capacitance is reassigned as line shunts at the original terminal bus ends of the two new line segments. The charging capacitance of the two new branch elements will be set to 0.

**CAPACITANCE:** The original charging capacitance of the line is converted approximately using percentage entered as PosAlongTheLine. In other words, the original capacitance will be scaled and assigned to the new segments as charging capacitance.

The default ShuntModel is LINEHUNTS.

**TreatAsMSLine:** Set to YES or NO. YES means that a new multi-section line will be created that is comprised of the two new lines. NO means that the two new lines will be controlled independently of each other.

**InterfacesAutoInsert(Type, DeleteExisting, UseDisplayFilters, Prefix, Limits);**

Use this action to automatically insert a group of interfaces. The new of the new interfaces defaults to "Area1-Area2" or "Zone1-Zone2" with an optional **Prefix**.

**Type:** The type of interfaces to insert. Choices are:

**AREA:** Area-to-area interfaces join adjacent areas (those that share at least one tie line).

**ZONE:** Zone-to-zone interfaces join adjacent interfaces (those that share at least one tie line).

**DeleteExisting:** Set to YES or NO. YES means that previously defined interfaces will be deleted before the new interfaces are automatically inserted. NO means automatically inserted interfaces will be added to the list of previously defined interfaces. If this parameter is not specified, then YES is assumed.

**UseDisplayFilters:** Set to YES or NO. YES means that only Areas and Zones with their Area/Zone filters set to YES will be used when automatically inserting interfaces. NO specifies all Areas and Zones will be used to automatically insert interfaces regardless of their Area/Zone filters. If this parameter is not specified, then NO is assumed.

**Prefix:** An optional prefix of up to three characters. Use this prefix to avoid duplicating names, particularly when some of the areas or zones have the same name. If this parameter is not specified, then "" is assumed.

**Limits:** Specifies the way the interfaces rating are set. Choices are:

AUTO: The interface rating is calculated based on the ratings of the components included in the interface.

ZEROS: All the interface limits are set to zero.

[value1, ..., value8]: The set of rating values to be used for the interfaces. The eight values must be specified, separated by commas.

The default parameter is ZEROS.

---

## Script Run Mode Actions

---

The following script commands are available during any of the submodes of Run Mode:

Animate;

CalculatePTDF	([transactor seller], [AREA num] [ZONE num] [SUPERAREA "name"] [INJECTIONGROUP "name"] [BUS num] [SLACK]	[transactor buyer], [AREA num] [ZONE num] [SUPERAREA name] [INJECTIONGROUP name] [BUS num] [SLACK]	LinearMethod) AC DC DCPS
---------------	---	---	-----------------------------------

CalculatePTDFMultipleDirections	(StoreValuesForBranches, YES or NO	StoreValuesForInterfaces, YES or NO	LinearMethod) AC DC DCPS
---------------------------------	---------------------------------------	--	-----------------------------------

CalculateLODF	([BRANCH nearbusnum farbusnum ckt],	LinearMethod);
---------------	--	----------------

CalculateTLR	([flowelement], [INTERFACE "name"]  [BRANCH nearbusnum farbusnum ckt]	direction, BUYER  SELLER	[transactor], same as above for PTDFs	LinearMet
--------------	---	-----------------------------------	---	-----------

CalculateTLRMultipleElement	(TypeElement  BRANCH  INTERFACE  BOTH	WhichElement, SELECTED OVERLOAD CTGOVERLOAD	direction, BUYER SELLER	[transactor], same as above for PTDFs	LinearMe
-----------------------------	---	--	-------------------------------	--	----------

CalculateVoltSense	([BUS num]);			
CalculateFlowSense	([flowelement], [INTERFACE "name"] [BRANCH busnum1 busnum2 ckt]	FlowType);  MW MVAR MVA		
CalculateLossSense	(FunctionType);  NONE  ISLAND  AREA  AREASA  SELECTED			
CalculateVoltToTransferSense	([transactor seller], [AREA num] [ZONE num] [SUPERAREA "name"] [INJECTIONGROUP "name"] [BUS num] [SLACK]	[transactor buyer], [AREA num] [ZONE num] [SUPERAREA name] [INJECTIONGROUP name] [BUS num] [SLACK]	TransferType [P, Q, or PQ]	TurnOffAV YES or NO
CalculateVoltSelfSense	(filter);			
SetSensitivitiesAtOutOfServiceToClosest;				
ZeroOutMismatches;				

### **Animate(DoAnimate);**

Use this action to animate all the open online diagrams.

**DoAnimate:** Set to YES or NO. YES means to start the animation of the open online diagrams, while NO means that the animation will be paused.

**CalculatePTDF([transactor seller], [transactor buyer], LinearMethod);**

Use this action to calculate the PTDF values between a seller and a buyer. You may optionally specify the linear calculation method. Note that the buyer and seller must not be same thing. If no Linear Method is specified, Lossless DC will be used.

**[transactor seller]:** The seller (or source) of power. There are six possible settings:

[AREA num], [AREA "name"], [AREA "label"]  
 [ZONE num], [ZONE "name"], [ZONE "label"]  
 [SUPERAREA "name"], [SUPERAREA "label"]  
 [INJECTIONGROUP "name"], [INJECTIONGROUP "label"]  
 [BUS num], [BUS "name\_nomkv"], [BUS "label"]  
 [SLACK]

**[transactor buyer]:** The buyer (or sink) of power. There are six possible settings, which are the same as for the seller.

**LinearMethod:** The linear method to be used for the PTDF calculation. The options are:

AC: for calculation including losses.  
 DC: for lossless DC.  
 DCPS: for lossless DC that takes into account phase shifter operation.

**CalculatePTDFMultipleDirections(StoreValuesForBranches, StoreValuesForInterfaces, LinearMethod);**

Use this action to calculate the PTDF values using multiple directions. Only directions with the field INCLUDE set to YES will be analyzed when calculating the multiple direction PTDF's. You may optionally specify the linear calculation method. If no Linear Method is specified, Lossless DC will be used.

**StoreValuesForBranches:** Set to YES or NO. YES means that the PTDF's for lines/transformers will be stored.

**StoreValuesForInterfaces:** Set to YES or NO. YES means that the PTDF's for interfaces will be stored.

**LinearMethod:** The linear method to be used for the PTDF calculation. The options are:

AC: for calculation including losses.  
 DC: for lossless DC.  
 DCPS: for lossless DC that takes into account phase shifter operation.

**CalculateLODF([BRANCH nearbusnum farbusnum ckt], LinearMethod);**

Use this action to calculate the Line Outage Distribution Factors (or the Line Closure Distribution Factors) for a particular branch. If the branch is presently closed, then the LODF values will be calculated, otherwise the LCDF values will be calculated. You may optionally specify the linear calculation method as well. If no Linear Method is specified, Lossless DC will be used.

**[BRANCH nearbusnum farbusnum ckt]:** The branch whose status is being changed. Can also use strings

```
[BRANCH "nearbusname_kv" "farbusname_kv" ckt]
[BRANCH "nearbuslabel" "farbuslabel" ckt]
[BRANCH "label"]
```

**LinearMethod:** The linear method to be used for the LODF calculation. The options are:

DC: for lossless DC.  
 DCPS: for lossless DC that takes into account phase shifter operation.  
 Note: AC is NOT an option for the LODF calculation.

**CalculateTLR([flow element], direction, [transactor], LinearMethod);**

Use this action to calculate the TLR values a particular flow element (transmission line or interface). You also specify one end of the potential transfer direction. You may optionally specify the linear calculation method. If no Linear Method is specified, Lossless DC will be used.

**[flow element]:** This is the flow element we are interested in. Choices are:

```
[INTERFACE "name"]
[INTERFACE "label"]
[BRANCH nearbusnum farbusnum ckt]
[BRANCH "nearbusname_kv" "farbusname_kv" ckt]
[BRANCH "nearbuslabel" "farbuslabel" ckt]
[BRANCH "label"]
```

**direction:** the type of the transactor. Either BUYER or SELLER.

**[transactor buyer]:** the transactor of power. There are six possible settings.

```
[AREA num], [AREA "name"], [AREA "label"]
[ZONE num], [ZONE "name"], [ZONE "label"]
[SUPERAREA "name"], [SUPERAREA "label"]
[INJECTIONGROUP "name"], [INJECTIONGROUP "label"]
[BUS num], [BUS "name_nomkv"], [BUS "label"]
[SLACK]
```

**LinearMethod:** The linear method to be used for the calculation. The options are:

AC: for calculation including losses.  
 DC: for lossless DC.  
 DCPS: for lossless DC that takes into account phase shifter operation.

**CalculateTLRMultipleElement(TypeElement, WhichElement, direction, [transactor], LinearMethod);**

Use this action to calculate the TLR values a multiple elements similar to as is done on the TLR multiple elements dialog. You also specify one end of the potential transfer direction. You may optionally specify the linear calculation method. If no Linear Method is specified, Lossless DC will be used.

**TypeElement :** May be either INTERFACE, BRANCH, or BOTH



**WhichElement** : There are three choices which represent which elements of the TypeElement specified will have TLR calculations performed.

SELECTED : Only branches or interfaces with their Selected Field = YES will be used.

OVERLOAD : Only branches that are presently overloaded using their normal ratings will be used

CTGOVERLOAD : You must have first run the contingency analysis. A branch or interface is included in the calculation if it has been overloaded during at least one contingency.

**direction**: the type of the transactor. Either BUYER or SELLER.

**[transactor buyer]**: the transactor of power. There are six possible settings.

[AREA num], [AREA "name"], [AREA "label"]

[ZONE num], [ZONE "name"], [ZONE "label"]

[SUPERAREA "name"], [SUPERAREA "label"]

[INJECTIONGROUP "name"], [INJECTIONGROUP "label"]

[BUS num], [BUS "name\_nomkv"], [BUS "label"]

[SLACK]

**LinearMethod**: The linear method to be used for the calculation. The options are:

AC: for calculation including losses.

DC: for lossless DC.

DCPS: for lossless DC that takes into account phase shifter operation.

**CalculateVoltSense([BUS num]);**

This calculates the sensitivity of a particular buses voltage to real and reactive power injections at all buses in the system. (Note: this assumes that the power is injected at a given bus and taken out at the slack bus).

**[BUS num]**: The bus to calculate sensitivities for.

**CalculateFlowSense([flow element], FlowType);**

This calculates the sensitivity of the MW, MVAR, or MVA flow of a line or interface to real and reactive power injections at all buses in the system. (Note: this assumes that the power is injected at a given bus and taken out at the slack bus).

**[flow element]**: This is the flow element we are interested in. Choices are:

[INTERFACE "name"]

[INTERFACE "label"]

[BRANCH busnum1 busnum2 ckt]

[BRANCH "name\_kv1" "name\_kv2" ckt]

[BRANCH "buslabel1" "buslabel2" ckt]

[BRANCH "label"]

**FlowType**: The type of flow to calculate this for. Either MW, MVAR, or MVA.

**CalculateLossSense(FunctionType);**

This action calculates the sensitivity of a real power loss function, PLosses, to bus real and reactive power injections. Stated mathematically, this action calculates  $d \text{ PLosses} / d P_i$  and  $d \text{ PLosses} / d Q_i$

$P_{losses}/d Q_i$ , where  $P_i$  and  $Q_i$  are the real and reactive power injections at bus  $i$ , respectively. Stated less formally, this action indicates how losses would change if one more MW or Mvar of power were injected at bus  $i$ .

**FunctionType:** Specifies how the losses are computed. Choices are:

NONE: No losses are calculated because a loss function is not specified.

ISLAND: Losses are calculated with respect to the losses in bus' island. If the power system consists of only one island, losses are computed with respect to the total system losses.

AREA: Losses are calculated with respect to the total losses for the area containing bus  $i$ . This is probably the most common loss function because usually one is concerned with minimizing losses for a particular area rather than for the entire case.

AREASA: Losses are calculated with respect to the total losses for the area containing bus  $i$  if bus  $i$  does not belong to a super area, and with respect to the total losses for the super area containing bus  $i$  if bus  $i$  does belong to a super area.

SELECTED: Losses are calculated with respect to the total losses for a group of areas whose *selected* field is set to YES.

**CalculateVoltToTransferSense([transactor seller], [transactor buyer], TransferType, TurnOffAVR);**

This calculates the sensitivity of bus voltage to a real or reactive power transfer between a seller and a buyer. The sensitivity is calculated for all buses in the system.

**[transactor seller]:** This is the seller (or source) of power. There are six possible settings:

[AREA num], [AREA "name"], [AREA "label"]

[ZONE num], [ZONE "name"], [ZONE "label"]

[SUPERAREA "name"], [SUPERAREA "label"]

[INJECTIONGROUP "name"], [INJECTIONGROUP "label"]

[BUS num], [BUS "name\_nomkv"], [BUS "label"]

[SLACK]

**[transactor buyer]:** This is the buyer (or sink) of power. There are six possible settings, which are the same as for the seller.

**TransferType:** The type of power transfer. The options are:

P – real power transfer

Q – reactive power transfer

PQ – both real and reactive power transfer. (Note: Real and reactive power transfers are calculated independently, but both are calculated.)

**TurnOffAVR:** Set to YES or NO. Set to YES to turn off AVR control for generators participating in the transfer. Set to NO to leave the AVR control unchanged for generators participating in the transfer.

**CalculateVoltSelfSense(filter);**

This calculates the sensitivity of a particular bus' voltage to real and reactive power injections at the same bus. (Note: This assumes that the power is injected at a given bus and taken out at the slack bus.)

**filter:** There are four options for the filter:

**CalculateVoltSelfSense**; : No filter specified means to merge all the buses of the system.

**CalculateVoltSelfSense("filtername")**; : "filtername" means to merge those buses that meet the filter.

**CalculateVoltSelfSense(AREAZONE)**; : AREAZONE means to merge those buses that meet the area/zone filters.

**CalculateVoltSelfSense(SELECTED)**; : SELECTED means to merge those buses whose *selected* field is set to YES.

#### **SetSensitivitiesAtOutOfServiceToClosest;**

This will take the P Sensitivity and Q Sensitivity values calculated using CalculateTLR, CalculateFlowSense, or CalculateVoltSense actions and then populate the respective values at out-of-service buses so that they are equal to the value at the closest in service bus. The "distance" to the in-service buses will be measured by the number of nodes. If an out-of-service bus is equally close to a set of buses, then the average of that set of buses will be used.

#### **ZeroOutMismatches;**

With this script command, the bus shunts are changed at each bus that has a mismatch greater than the MVA convergence tolerance so that the mismatch at that bus is forced to zero.

In addition to the general run mode actions, there are specific actions for the following submodes: [PowerFlow](#), [Contingency](#), [ATC](#), [Fault](#), [PV](#) and [QV](#).

---

## Script PowerFlow Submode Actions

---

The following actions are specific to the Run Mode submode PowerFlow:

```
DoCTGAction([contingency action]);
SolvePowerFlow(SolMethod, "filename1", "filename2", CreateIfNotFound1,
CreateIfNotFound2);
ResetToFlatStart(FlatVoltagesAngles, ShuntsToMax, LTCsToMiddle, PSAnglesToMiddle);
SolvePrimalLP("filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);
InitializePrimalLP("filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);
SolveSinglePrimalLPOuterLoop("filename1", "filename2", CreateIfNotFound1,
CreateIfNotFound2);
SolveFullSCOPF(BCMethod, "filename1", "filename2", CreateIfNotFound1,
CreateIfNotFound2);
OPFWriteResultsAndOptions ("filename");
DiffFlowSetAsBase;
DiffFlowClearBase;
DiffFlowMode (diffmode);
    PRESENT
    BASE
    DIFFERENCE
```

### **DoCTGAction([contingency action]);**

Call this action to use the formats seen in the CTGElement subdata record for Contingency Data. Note that all actions are supported, except COMPENSATION sections are not allowed.

### **SolvePowerFlow(SolMethod, "filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);**

Call this action to perform a single power flow solution. The parameters are all optional and specify a conditional response depending on whether the solution is successfully found. If parameters are not passed then default values will be used.

**SolMethod** The solution method to be used for the power flow calculation. The options are:

- RECTNEWT: For Rectangular Newton-Raphson method.
- POLARNEWTON: For Polar Newton-Raphson method.
- GAUSSSEIDEL: For Gauss-Seidel method.
- FASTDEC: For Fast Decoupled method.
- DC: For DC method.
- ROBUST: For ROBUST method.

The default method is RECTNEWT.

**"filename1"** The filename of the auxiliary file to be loaded if there is a successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

**"filename2"** The filename of the auxiliary file to be loaded if there is a NOT successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

**CreateIfNotFound1** Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename1. Default Value is NO.

**CreateIfNotFound2** Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename2. Default Value is NO.

### **ResetToFlatStart (FlatVoltagesAngles, ShuntsToMax, LTCsToMiddle, PSAnglesToMiddle);**

Use this action to initialize the Power Flow Solution to a "flat start." The parameters are all optional and specify a conditional response depending on whether the solution is successfully found. If parameters are not passed then default values will be used.

**FlatVoltagesAngles** Set to YES or NO. YES means setting all the voltage magnitudes and generator setpoint voltages to 1.0 per unit and all the voltage angles to zero. Default Value is YES.

**ShuntsToMax** Set to YES or NO. YES means to increase Switched Shunts Mvar half way to maximum. Default Value is NO.

**LTCsToMiddle** Set to YES or NO. YES means setting the LTC Transformer Taps to middle of range. Default Value is NO.

**PSAnglesToMiddle** Set to YES or NO. YES means setting Phase Shifter angles to middle of range. Default Value is NO.

### **SolvePrimalLP("filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);**

Call this action to perform a primal LP OPF solution. The parameters are all optional and specify a conditional response depending on whether the solution is successfully found. If parameters are not passed then default values will be used.

**"filename1"** The filename of the auxiliary file to be loaded if there is a successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

**"filename2"** The filename of the auxiliary file to be loaded if there is a NOT successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

**CreateIfNotFound1** Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename1. Default Value is NO.

**CreateIfNotFound2** Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename2. Default Value is NO.

### **SolveSinglePrimalLPOuterLoop("filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);**

This action is basically identical to the SolvePrimalLP action, except that this will only perform a single optimization. The SolvePrimalLP will iterate between solving the power flow and an optimization until this iteration converges. This action will only solve the optimization routine once, then resolve the power flow once and then stop.

**InitializePrimalLP("filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);**

This command clears all the structures and results of previous primal LP OPF solutions. The parameters are all optional and specify a conditional response depending on whether the solution is successfully found. If parameters are not passed then default values will be used.

**"filename1"** The filename of the auxiliary file to be loaded if there is a successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

**"filename2"** The filename of the auxiliary file to be loaded if there is a NOT successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

**CreateIfNotFound1** Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename1. Default Value is NO.

**CreateIfNotFound2** Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename2. Default Value is NO.

**SolveFullSCOPF (BCMMethod, "filename1", "filename2", CreateIfNotFound1, CreateIfNotFound2);**

Call this action to perform a full Security Constrained OPF solution. The parameters are all optional and specify a conditional response depending on whether the solution is successfully found. If parameters are not passed then default values will be used.

**BCMMethod** The solution method to be used for solving the base case. The options are:

POWERFLOW – for single power flow algorithm.

OPF – for the optimal power flow algorithm.

Default Value is POWERFLOW.

**"filename1"** The filename of the auxiliary file to be loaded if there is a successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

**"filename2"** The filename of the auxiliary file to be loaded if there is a NOT successful solution. You may also specify STOP, which means that all AUX file execution should stop under the condition. Default Value is "".

**CreateIfNotFound1** Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename1. Default Value is NO.

**CreateIfNotFound2** Set to YES or NO. YES means that objects which can not be found will be created while reading in DATA sections of filename2. Default Value is NO.

**OPFWriteResultsAndOptions("filename");**

Writes out all information related to OPF analysis as an auxiliary file. This includes Limit Monitoring Settings, options for Areas, Buses, Branches, Interfaces, Generators, SuperAreas, OPF Solution Options.

**DiffFlowSetAsBase;**

Call this action to set the present case as the base case for the difference flows abilities of Simulator.

**DiffFlowClearBase;**

Call this action to clear the base case for the difference flows abilities of Simulator.

**DiffFlowMode(diffmode);**

Call this action to change the mode for the difference flows abilities of Simulator.

**diffmode:**

String that starts with 'P' changes it to PRESENT

String that starts with 'B' changes it to BASE

String that starts with 'D' changes it to DIFFERENCE

---

## Script Contingency Submode Actions

---

The following actions are specific to the Run Mode Submode Contingency:

```
CTGSolveAll;
CTGSolve("ContingencyName");
CTGSetAsReference;
CTGRestoreReference;
CTGProduceReport("filename");
CTGWriteResultsAndOptions("filename");
CTGAutoInsert;
CTGCalculateOTDF( [transactor seller], [transactor buyer], LinearMethod);
[AREA num] [AREA num] AC
[ZONE num] [ZONE num] DC
[SUPERAREA "name"] [SUPERAREA "name"] DCPS
[INJECTIONGROUP "name"] [INJECTIONGROUP "name"]
[BUS num] [BUS num]
[SLACK] [SLACK]
```

### **CTGSolveAll;**

Call this action to solve all the contingencies which are not marked skip.

### **CTGSolve("ContingencyName");**

Call this action solve a particular contingency. The contingency is denoted by the "Contingency Name".

### **CTGSetAsReference;**

Call this action to set the present system state as the reference for contingency analysis.

### **CTGRestoreReference;**

Call this action to reset the system state to the reference state for contingency analysis.

### **CTGProduceReport("filename");**

Produces a text-based contingency analysis report using the settings defined in CTG\_Options.

### **CTGWriteResultsAndOptions("filename");**



Writes out all information related to contingency analysis as an auxiliary file. This includes Contingency Definitions, Limit Monitoring Settings, Contingency Results, Solution Options, CTG Options as well as any Model Criteria that are used by the Contingency Definitions.

**CTGAutoInsert;**

Automatically generates a contingency list using the settings defined in CTG\_Options.

**CTGCalculateOTDF([transactor seller], [transactor buyer], LinearMethod);**

This action calculate OTDFs for each contingency-caused branch or interface violation. The values calculated will be a measure of what percent of a transfer would appear on the branch or interface after the respective contingency occurs. Realize for branch violations, that the sign of the OTDF value will be relative to the direction of the MW flow found during the contingency analysis.

The parameters will be used to calculate the PTDFs for the transfer direction you are interested in. These PTDF values will then be used throughout the OTDF calculation. Note that the buyer and seller must not be same thing. If no Linear Method is specified, Lossless DC will be used.

NOTE: All the parameters are optional. If they are not specified, then the existing PTDFs will be used.

**[transactor seller]:** The seller (or source) of power. There are six possible settings:

- [AREA num]
- [ZONE num]
- [SUPERAREA "name"]
- [INJECTIONGROUP "name"]
- [BUS num]
- [SLACK]

**[transactor buyer]:** The buyer (or sink) of power. There are six possible settings, which are the same as for the seller.

**LinearMethod:** The linear method to be used for the PTDF calculation. The options are:

- AC: for calculation including losses.
- DC: for lossless DC.
- DCPS: for lossless DC that takes into account phase shifter operation.

---

## Script ATC Submode Actions

---

The following actions are specific to the Run Mode Submode ATC:

ATCDetermine	([transactor seller], [AREA num] [ZONE num] [SUPERAREA "name"] [INJECTIONGROUP "name"] [BUS num] [SLACK]	[transactor buyer]); [AREA num] [ZONE num] [SUPERAREA name] INJECTIONGROUP name] [BUS num] SLACK]
--------------	---	--

ATCRestoreInitialState;

ATCIncreaseTransferBy (amount);

ATCTakeMeToScenario (RL, G, I);

ATCDetermineATCFor (RL, G, I, ApplyTransfer);

ATCWriteResultsAndOptions ("filename");

ATCWriteToExcel ("worksheetname");

### **ATCDetermine([transactor seller], [transactor buyer]);**

Use this action to calculate the Available Transfer Capability (ATC) between a seller and a buyer. Note that the buyer and seller must not be same thing. Other options regarding ATC calculations should be set using a DATA section via the ATC\_Options object type.

**[transactor seller]:** The seller (or source) of power. There are six possible settings

[AREA num], [AREA "name"], [AREA "label"]  
[ZONE num], [ZONE "name"], [ZONE "label"]  
[SUPERAREA "name"], [SUPERAREA "label"]  
[INJECTIONGROUP "name"], [INJECTIONGROUP "label"]  
[BUS num], [BUS "name\_nomkv"], [BUS "label"]  
[SLACK]

**[transactor buyer]:** The buyer (or sink) of power. There are six possible settings which are the same as for the seller.

### **ATCRestoreInitialState;**

Call this action to restore the initial state for the ATC tool.

**ATCIncreaseTransferBy(amount);**

Call this action to increase the transfer between the buyer and seller.

**ATCTakeMeToScenario(RL, G, I);**

Call this action to set the present case according to Scenario RL, G, I.

**ATCDetermineATCFor(RL, G, I, ApplyTransfer);**

Call this action to determine the ATC for Scenario RL, G, I.

**ApplyTransfer:** Set this value to YES to leave the system state at the transfer level that was determined. When using the Iterated Linear then Full Contingency solution method, the system state will retain the transfer level but the contingency will not be applied.

**ATCWriteResultsAndOptions("filename");**

Writes out all information related to ATC analysis as an auxiliary file. This includes Contingency Definitions, Limit Monitoring Settings, Solution Options, ATC Options, ATC results, as well as any Model Criteria that are used by the Contingency Definitions.

**ATCWriteResultsAndOptions("worksheetname");**

Sends ATC analysis results to an Excel spreadsheet. This script command is available only for Multiple Scenarios ATC analysis.

**"worksheetname"** The name of the Excel sheet where the results will be sent to.

---

**Script Fault Submode Actions**


---

The following actions are specific to the Run Mode Submode Fault:

```

Fault    ([BUS num],                                faulttype,  R,    X);
Fault    ([BRANCH nearbusnum farbusnum ckt],        faultlocation, faulttype,  R,    X) ;
                                                SLG
                                                LL
                                                3PB
                                                DLG

```

**Fault([Bus num, faulttype, R, X]);**

**Fault([BRANCH nearbusnum farbusnum ckt], faultlocation, faulttype, R, X);**

Call this function to calculate the fault currents for a fault. If the fault element is a bus then do not specify the fault location parameter. If the fault element is a branch, then the fault location is required.

**[Bus num]:** This specifies the bus at which the fault occurs. You may also specify the bus using secondary keys or labels:

[BUS "name\_nomkv"]

[BUS "label"]

**[BRANCH nearbusnum farbusnum ckt]:** This specifies the branch on which the fault occurs. You may also specify the branch using secondary keys or labels:

[BRANCH "name\_kv1" "name\_kv2" ckt]

[BRANCH "buslabel1" "buslabel2" ckt]

[BRANCH "label"]

**Fault location:** This specifies the percentage distance along the branch where the fault occurs. This percent varies from 0 (meaning at the nearbus) to 100 (meaning at the far bus)

**Faulttype:** This specified the type of fault which occurs. There are four options:

SLG: Single Line To Ground fault

LL: Line to Line Fault

3PB: Three Phase Balanced Fault

DLG: Double Line to Group Fault.

**R, X:** These parameters are optional and specify the fault impedance. If none are specified, then a fault impedance of zero is assumed.

---

## Script PV Submode Actions

---

The following actions are specific to the Run Mode Submode PV:

PVCreate	("name",	[element source],	[element sink]);
		[INJECTIONGROUP "name"]	[INJECTIONGROUP "name"]
PVSetSourceAndSink	("name");	[element source],	[element sink]);
		[INJECTIONGROUP "name"]	[INJECTIONGROUP "name"]
PVRun	("name");		
PVClearResults	("name");		
PVStartOver	("name");		
PVDestroy	("name");		
PVWriteResultsAndOptions	("filename");		
RefineModel	(objecttype,	filter,	Action ForWhichValue);
	AREA		TRANSFORMERTAPS
	ZONE		SHUNTS
			OFFAVR

### **PVCreate("name", [elementSource], [elementSink]);**

Call the function to create a PV study with "name" as identifier. You may optionally specify the source and sink elements for the study transaction.

**"name"**: String that identifies the PV study to be created.

**[element source]**: The source of power for the PV study. There is only one possible setting:

[INJECTIONGROUP "name"] or [INJECTIONGROUP "label"]

**[element sink]**: The sink of power for the PV study. There is only one possible setting, which is the same as for the source.

### **PVSetSourceAndSink("name", [elementSource], [elementSink]);**

Call the function to specify the source and sink elements to perform the PV study called "name".

**"name"**: String that identifies the PV study for which the source and sink elements are to be assigned to.

**[element source]:** The source of power for the PV study. There is only one possible setting:

[INJECTIONGROUP "name"] or [INJECTIONGROUP "label"]

**[element sink]:** The sink of power for the PV study. There is only one possible setting, which is the same as for the source.

**PVRun("name");**

Call the function to start the PV study called "name".

**"name":** String that identifies the PV study.

**PVClearResults("name");**

Call the function to clear all the results of the PV study called "name".

**"name":** String that identifies the PV study.

**PVStartOver("name");**

Call the function to start over the PV study called "name". This includes clear the activity log, clear results, restore the initial state, set the current state as initial state, and initialize the step size.

**"name":** String that identifies the PV study.

**PVDestroy("name");**

Call the function to destroy the PV study called "name".

**"name":** String that identifies the PV study.

**PVWriteResultsAndOptions("filename");**

Call this action to save all the PV results and options in the auxiliary file "filename".

**RefineModel(objecttype, filter, Action, Tolerance);**

Call this function to refine the system model to fix modeling idiosyncrasies that cause premature loss of convergence during PV and QV studies.

**objecttype:** The objecttype being selected.

AREA

ZONE

**filter:** There are three options for the filter:

**RefineModel(..., "", ...);** No filter specified means to select all objects of this type.

**RefineModel(..., "filtername", ...);** "filtername" means select those that meet the filter.

**Action:** The way the model will be refined. Choices are:

TRANSFORMERTAPS Fix all transformer taps at their present values if their Vmax – Vmin is less than or equal to the user specified tolerance.

SHUNTS Fix all shunts at their present values if their Vmax – Vmin is less than or equal to the user specified tolerance.

OFFAVR Remove units from AVR control, thus locking their MVAR output at its present value if their  $Q_{\max} - Q_{\min}$  is less or equal to the user specified tolerance.

**Tolerance:** Tolerance value.

---

## Script QV Submode Actions

---

The following actions are specific to the Run Mode Submode QV:

```
QVRun      ("filename",      InErrorMakeBaseSolvable);
                                     YES or NO
```

NOTE: The QV study is always performed on selected buses.

```
QVWriteResultsAndOptions      ("filename");
```

```
RefineModel      (objecttype,      filter,      Action      ForWhichValue);
                                     AREA      TRANSFORMERTAPS
                                     ZONE      SHUNTS
                                     OFFAVR
```

### **QVRun("filename", InErrorMakeBaseSolvable);**

Call the function to start a QV study for the list of buses whose SELECTED? field is set to YES.

**"filename"**: This specifies the file to which to save a comma-delimited version of the results.

**InErrorMakeBaseSolvable**: This specifies whether to perform a solvability analysis of the base case if the pre-contingency base case can not be solved. If not specified, then YES is assumed.

### **QVWriteResultsAndOptions("filename");**

Call this action to save all the QV results and options in the auxiliary file "filename".

### **RefineModel(objecttype, filter, Action, Tolerance);**

Call this function to refine the system model to fix modeling idiosyncrasies that cause premature loss of convergence during PV and QV studies.

**objecttype**: The objecttype being selected.

AREA

ZONE

**filter**: There are three options for the filter:

**RefineModel(..., "", ...)**:: No filter specified means to select all objects of this type.

**RefineModel(..., "filtername", ...)**:: "filtername" means select those that meet the filter.

**Action**: The way the model will be refined. Choices are:



TRANSFORMERTAPS Fix all transformer taps at their present values if their  $V_{\max} - V_{\min}$  is less than or equal to the user specified tolerance.

SHUNTS Fix all shunts at their present values if their  $V_{\max} - V_{\min}$  is less than or equal to the user specified tolerance.

OFFAVR Remove units from AVR control, thus locking their MVAR output at its present value if their  $Q_{\max} - Q_{\min}$  is less or equal to the user specified tolerance.

**Tolerance:** Tolerance value.

---

**Data Section**

---

The Data Section of the script file format can contain numerous functions for modifying the data of a power system case. The DATA section begins with a left curly brace and ends with a right curly brace.

**DATA DataName (object\_type, [list\_of\_fields], file\_type\_specifier)**

```
{  
data_list_1  
.  
.  
.  
data_list_n  
}
```

Immediately following the DATA keyword, you may optionally include a DataName. By including the DataName, you can make use of the script command LoadData("filename", DataName) to call this particular data section from another auxiliary file.

The first line of the DATA section is referred to as the [Data Argument List](#). The [Data Argument List](#) defines the type and format of the data contained in the following section enclosed in curly braces.

Inside of each DATA section, [data lists](#) are given according to the data definition given in the [Data Argument List](#).

---

## Data Argument List

---

The DATA argument list identifies what the information section contains. A left and right parenthesis "(" ")" mark the beginning and end of the argument list.

The **file\_type\_specifier** parameter distinguishes the information section as containing custom auxiliary data (as opposed to Simulator's native auxiliary formats), and indicates the format of the data. Currently, the parser recognizes two values for **file\_type\_specifier**:

(blank) or AUXDEF	Data fields are space delimited
CSV, AUXCSV, or CSVAUX	Data fields are comma delimited

The **object\_type** parameter identifies the type of object or data element the information section describes or models. For example, if **object\_type** equals BUS, then the data describes BUS objects. Below is a list of many of the object types recognized by Simulator:

3WXFormer	DefDrawSuperArea	OPF_Options_and_Results
Area	DefDrawZone	OPFSolutionSummary
ATC_Options	Direction	Owner
ATCExtraMonitor	DynamicFormatting	PieSizeColorOptions
ATCGeneratorChange	Equiv_Options	PostPowerFlowActions
ATCInterfaceChange	ExportBidCurve	PTDF_Options
ATCLineChange	Filter	PVCurve_Options
ATCLineChangeB	GEMotor	PWCaseInformation
ATCScenario	Gen	PWFormOptions
ATCZoneChange	GeoDataViewStyle	PWLPOPFGTGViol
AuxFileExportFormatData	GlobalContingencyActions	PWLPTabRow
AuxFileExportFormatDisplay	GScenarioName	PWOPFTimePoint
BGCalculatedField	HintDefVAlues	PWPVResultListContainer
Branch	IG_AutoInsert_Options	QVCurve
Bus	ImportBidCurve	QVCurve_Options
BusGroupSwapRec	ImportExportBidCurve	RLScenarioName
BusNumberSwap	InjectionGroup	Scale_Options
BusViewFormOptions	Interface	Schedule
CaseInfo_Options	InterfaceAEElement	ScreenLayer
ColorMap	InterfaceBEElement	SelectByCriteriaSet
Contingency	IScenarioName	ShapefileExportDescription
Ctg_AutoInsert_Options	Limit_Monitoring_Options	ShortCut
Ctg_Options	LimitCost	Shunt
CTGElementBlock	LimitSet	Sim_Environment_Options
CustomCaseInfo	Load	Sim_Misc_Options
CustomCaseInfoRow	LODF_Options	SimSimulation_Options
CustomColors	LPOPFMarginalControls	Sim_Solution_Options
CustomExpression	LPVariable	StudyMWTransactions

DataGrid	MessLog_Options	Substation
DCTransmissionLine	ModelCondition	SuperArea
DefDrawArea	ModelExpression	TLR_Options
DefDrawBackground	ModelFilter	TransferLimiter
DefDrawBus	MTDCRecord	Transformer
DefDrawGen	MultiSectionLine	TSSchedule
DefDrawInterface	MvarMarginalCostValues	UC_Options
DefDrawLineTransformer	MWMarginalCostValues	ViolationCTG
DefDrawLoad	MWTransaction	WhatOccurredDuringContingency
DefDrawShunt	Nomogram	XFCorrection
DefDrawSubstation	NomogramInterface	Zone

The list of object types Simulator's auxiliary file parser can recognize will grow as new applications for the technology are found. Within Simulator, you will always be able to obtain a list of the available **object\_types** by going to the **Auxiliary Files** ribbon group on the [Window](#) ribbon tab and choosing **Export Object Fields**.

The **list\_of\_fields** parameter lists the types of values the ensuing records in the data section contain. The order in which the fields are listed in **list\_of\_fields** dictates the order in which the fields will be read from the file. Simulator currently recognizes over 2,000 different field types, each identified by a specific field name. Because the available fields for an object may grow as new applications are developed, you will always be able to obtain a list of the available **object\_types** by going to the **Auxiliary Files** ribbon group on the [Window](#) ribbon tab and choosing **Export Object Fields**. Certainly, only a subset of these fields would be found in a typical custom auxiliary file. In crafting applications to export custom auxiliary files, developers need concern themselves only the fields they need to communicate between their applications and Simulator. A few points of interest regarding the **list\_of\_fields** are:

- The **list\_of\_fields** may take up several lines of the text file.
- The **list\_of\_fields** should be encompassed by braces [ ].
- When encountering the PowerWorld comment string `'''` in one of these lines of the text file, all text to the right of this is ignored.
- Blank lines, or lines whose first characters are `'''` will be ignored as comments.
- Field names must be separated by commas.

#### Example DATA Argument List

```
DATA (BUS, [BusNomKV, Bus, // comment here
ABCPhaseAngle: 1, ABCPhaseAngle: 2, ABCPhaseV, ABCPhaseV: 1,
// comments allowed here to

// note that blank rows are ignored
AreaNum, BusAngle, BusB, BusCat, BusEquiv, BusG,
BusGenericSensV, BusKVVolt, BusLambda, BusLoadMVA, // more comment
BusLoadMW, BusLongName])
```

One general note regarding the field names however. Some field names may be augmented with a field location. One example of this is the field LineMW. For a branch, there are two MW flows associated with the line: one MW flow at the from bus, and one MW flow at the to bus. So that the number of fields does not become huge, the same field name is used for both of these values. For the from bus flow, we write LineMW: 0, and for the to bus flow, we write LineMW: 1. Note that fieldnames such as LineMW: 0 may simply leave off the: 0.



---

## Data Key Fields

---

Simulator uses certain fields to identify the specific object being described. These fields are called key fields. For example, the key field for BUS objects is BusNum, because a bus can be identified uniquely by its number. The key fields for GEN objects are BusNum and GenID. To properly identify each object, the object's key fields must be present. They can appear in any order in the **list\_of\_fields** (i.e. they need not be the first fields listed in **list\_of\_fields**). As long as the key fields are present, Simulator can identify the specific object. By going to the **Auxiliary Files** ribbon group on the [Window](#) ribbon tab and choosing **Export Case Object Fields** you will obtain a list of fields available for each object type. In this output, the key fields will appear with asterisks \*.

---

**Data List**

---

After the data argument list is completed, the Data list is given. The data section lists the values of the fields for each object in the order specified in **list\_of\_fields**. The data section begins with a left curly brace and ends with a right curly brace. A few points of interest regarding the **value\_list**:

- The **value\_list** may take up several lines of the text file.
- Each new data object must start on its own line of text.
- When encountering the PowerWorld comment string `'//'` in one of these lines of the text file, all text to the right of this is ignored.
- Blank lines, or lines whose first characters are `'//'` will be ignored as comments.
- Remember that the right curly brace must appear on its own line at the end of the **data\_list**.
- If the **file\_type\_specifier** is CSV, the values should be separated by commas. Otherwise, separate the field names using spaces.
- Strings can be enclosed in double quotes, but this is not required. You should however always enclose strings that contain spaces (or commas) in quotes. Otherwise, strings containing commas would cause errors for comma-delimited files, and spaces would cause errors for space-delimited formatted files.

---

## Data SubData Sections

---

The format described in the [Data Section](#) works well for most kinds of data in Simulator. It does not work as well however for data that stores a list of objects. For example, a contingency stores some information about itself (such as its name), and then a list of contingency elements, and possible a list of limit violations as well. For data such as this, Simulator allows `<SubData>`, `</SubData>` tags that store lists of information about a particular object. This formatting looks like the following:

**DATA (object\_type, [list\_of\_fields], file\_type\_specifier)**

```
{
value_list_1
  <SUBDATA subobject_type1>
    precise format describing an object_type1
    precise format describing an object_type1
    .
    .
    .
  </SUBDATA>
  <SUBDATA subobject_type2>
    precise format describing an object_type2
    precise format describing an object_type2
    .
    .
    .
  </SUBDATA>
value_list_2
.
.
.
value_list_n
}
```

Note that the information contained inside the `<SubData>`, `</SubData>` tags may not be flexibly defined. It must be written in a precisely defined order that will be documented for each SubData type.



---

**Data ATC\_Options**


---

**RLScenarioName****GScenarioName****IScenarioName**

These three sections contain the pretty names of the RL Scenarios, G Scenarios, and I Scenarios. Each line consists of two values: Scenario Number and a name string enclosed in quotes.

**Scenario Number:** The scenarios are number 0 through the number of scenarios minus 1.

**Scenario Name:** These represent the names of the various scenarios.

**Example:**

```
<SUBDATA RLScenarioName>
//Index Name
0 "Scenario Name 0"
1 "Scenario Name 1"
</SUBDATA>
```

**ATCMemo**

This section contains the memo text for the ATC analysis.

**Example:**

```
<SUBDATA ATCMemo>
//Memo
"Comments for the ATC analysis"
</SUBDATA>
```

---

## Data ATCExtraMonitor

---

### ATCFlowValue

This subdata section contains a list of a flow values for specified transfer levels. Each line consists of two values: Flow Value (flow on the monitored element) and a Transfer Level (in MW).

**Flow Value:** Contains a string describing which monitor this belongs to.

**Transfer Level:** Contains the value for this extra monitor at the last linear iteration.

### Example:

```
<SUBDATA ATCFlowValue>
//MWFlow TransferLevel
94.05 55.30
105.18 80.58
109.02 107.76
</SUBDATA>
```

---

**Data ATCScenario**


---

**TransferLimiter**

This subdata section contains a list of the TransferLimiters for this scenario. Each line contains fields relating one of the Transferlimiters. The fields are written out in the following order:

**Limiting Element:** Contains a description of the limiting element. The possible values are

"PowerFlow Divergence"  
 "AREA num"  
 "SUPERAREA name"  
 "ZONE num"  
 "BRANCH num1 num2 ckt"  
 "INJECTIONGROUP name"  
 "INTERFACE name"

**Limiting Contingency:** The name of the limiting contingency. If blank, then this means it's a limitation in the base case.

**MaxFlow:** The transfer limitation in MW.

**PTDF:** The PTDF on the limiting element in the base case (not in percent).

**OTDF:** The OTDF on the limiting element under the limiting contingency.

**LimitUsed:** The limit which was used to determine the MaxFlow in per unit.

**PreTransEst:** The estimated flow on the line after the contingency but before the transfer in per unit.

**MaxFlowAtLastIteration:** The total transfer at the last iteration in per unit.

**IterativelyFound:** Either YES or NO depending on whether it was iteratively determined.

**Example:**

```
<SUBDATA TransferLimiter >
"BRANCH 40767 42103 1" "Contin" 2.84 -0.0771 -0.3883 -4.35 -4.35 -0.01 "-55.88"
YES
"BRANCH 42100 42321 1" "Contin" 4.42 0.1078 0.5466 6.50 5.64 1.57 " 22.59" NO
"BRANCH 42168 42174 1" "Contin" 7.45 -0.0131 -0.0651 -1.39 -1.09 4.60 "-33.31" NO
"BRANCH 42168 42170 1" "Contin" 8.54 0.0131 0.0651 1.39 1.02 5.69 " 26.10" NO
"BRANCH 41004 49963 1" "Contin" 9.17 -0.0500 -0.1940 -4.39 -3.16 6.32 " 68.73" NO
"BRANCH 46403 49963 1" "Contin" 9.53 0.0500 0.1940 4.46 3.16 6.68 "-68.68" NO
"BRANCH 42163 42170 1" "Contin" 10.14 -0.0131 -0.0651 -1.39 -0.92 7.29 "-15.58" NO

</SUBDATA>
```

**ATCExtraMonitor**

This subdata section contains a list of the ATCExtraMonitors for this scenario. Each line contains three fields relating one of the ATCExtraMonitors. The first field describes the

ATCExtraMonitor which this subdata corresponds to. The second and third variables are the initial value and sensitivity for this extra monitor for the scenario. An option fourth field may be included if we are using one of the iterated ATC solution options. This field must be the String "ATCFlowValue".

**Monitor Description:** Contains a string describing which monitor this belongs to.

**Initial Value:** Contains the value for this extra monitor at the last linear iteration.

**Sensitivity:** Contains the sensitivity of this monitor.

**ATCFlowValue:** A string which signifies that a block will follow which stores a list of flow values for specified transfer levels. Each line of the block consists of two values: Flow Value (flow on the monitored element) and a Transfer Level (in MW). The block is terminated when a line of text that starts with 'END' is encountered.

**Example:**

```
<SUBDATA ATCExtraMonitor>
"Interface<KEY1>Left-Right</KEY1>" 40.0735 0.633295
"Branch<KEY1>2</KEY1><KEY2>5</KEY2><KEY3>1</KEY3>" 78.7410 0.266589
</SUBDATA>
```

---

**Data AuxFileExportFormatData**


---

**DataBlockDescription**

This subdata section is used to define the objects that should be included in an auxiliary file along with their fields, subdata sections, and any filter used to specify which objects should be included. Each line contains the following:

**ObjectType:** Name of the object to include in the auxiliary file.

**[FieldList]:** List of fields to include. Must be enclosed in brackets. This list can either be space-delimited or comma-delimited.

**[SubdataList]:** List of subdata sections to include. This list must be enclosed in brackets and can be either space-delimited or comma-delimited. Include empty brackets to not include subdata or for objects that do not have any subdata sections.

**"Filter":** Description of the filter to use for determining which objects to include. This must be enclosed in double quotes. If no filter is to be used, empty double quotes should be included. Valid entries are: "", "filtername", "AREAZONE", and "SELECTED".

**Example:**

```
<SUBDATA DataBlockDescription>
// ObjectType [FieldList] [SubdataList] "Filter"
Area [AreaName, AreaNum] [] "SELECTED"
Gen [BusNum, BusName, GenID] [BidCurve, ReactiveCapability] ""
</SUBDATA>
```

---

## Data AuxFileExportFormatDisplay

---

### DataBlockDescription

This subdata section is used to define the objects that should be included in an auxiliary file along with their fields, subdata sections, and any filter used to specify which objects should be included. Each line contains the following:

**ObjectType:** Name of the object to include in the auxiliary file.

**[FieldList]:** List of fields to include. Must be enclosed in brackets. This list can either be space-delimited or comma-delimited.

**[SubdataList]:** List of subdata sections to include. This list must be enclosed in brackets and can be either space-delimited or comma-delimited. Include empty brackets to not include subdata or for objects that do not have any subdata sections.

**"Filter":** Description of the filter to use for determining which objects to include. This must be enclosed in double quotes. If no filter is to be used, empty double quotes should be included. Valid entries are: "", "filtername", "AREAZONE", and "SELECTED".

#### Example:

```
<SUBDATA DataBlockDescription>
// ObjectType [FieldList] [SubdataList] "Filter"
DisplayArea [AreaName, AreaNum, SOAuxiliaryID] [] ""
DisplayTransmissionLine [BusNum, BusNum:1, LineCircuit, SOAuxiliaryID] [Line]
"Nominal Voltage > 138 kV"
</SUBDATA>
```

---

**Data BGCalculatedField**

---

**Condition**

Bus Group Calculated Fields are used for creating a calculation that operates on the buses in a area, zone, or substation. Part of the definition is a filter which specifies which buses to operate over. This subdata section is identical to the Condition subdata section of the Filter object type.

---

**Data Bus**

---

**MWMarginalCostValues****MvarMarginalCostValues****LPOPFMarginalControls**

These three sections contain specific values computed for an OPF solution. In MWMarginalCostValues or MvarMarginalCostValues these specific values are the MW or Mvar marginal prices for each constraint. In LPOPFMarginalControls the values are the sensitivities of the controls with respect to the cost of each bus.

**Example:**

```
<SUBDATA MWMarginalCostValues>  
//Value  
16.53  
0.00  
21.80  
</SUBDATA>
```



---

**Data BusViewFormOptions**


---

**BusViewBusField****BusViewFarBusField****BusViewGenField****BusViewLineField****BusViewLoadField****BusViewShuntField**

The values represent specific fields on the custom defined bus view onelines. Each line contains two values:

**Location** : The various locations on the customized bus view contain slots for fields. This is the slot number.

**FieldDescription** : This is a string enclosed in double quotes. The string itself is delimited by the @ character. The string contains five values:

Name of Field: The name of the field. Special fields that appear on dialog by default have special names. Otherwise these are the same as the fieldnames of the AUX file format (for the "other fields" feature on the dialogs).

Total Digit: Number of total digits for a numeric field.

Decimal Points: Number of decimal points for a numeric field.

Color: This is the color of the field. It is not presently used.

Increment Value: This is the "delta per mouse" click for the field.

**Example:**

```
<SUBDATA BusViewLineField>
0 "MW Flow@6@1@0@0"
1 "MVar Flow@6@1@0@0"
2 "MVA Flow@6@1@0@0"
3 "BusAngle:1@6@2@0@0"
</SUBDATA>
```

---

## Data ColorMap

---

### ColorPoint

A colorpoint is simply described by a real number (between 0 and 100) and an integer describing the color written on a single line of text

**cmvalue cmcolor (Example: 100 56648)**

**cmvalue:** Real number between 0 and 100 (minimum to maximum value).

**cmcolor:** Integer between 0 and 16,777,216. Value is determined by taking the red, green and blue components of the color and assigning them a value between 0 and 255. The color is then equal to  $\text{red} + 256 * \text{green} + 256 * 256 * \text{blue}$ .

#### Example:

```
<SUBDATA ColorPoint>
// Value Color
100.0000 127
87.5000 255
62.5000 65535
50.0000 8388479
37.5000 16776960
12.5000 16711680
0.0000 8323072
</SUBDATA>
```

---

## Data Contingency

---

### CTGElement

A contingency element is described by up to four entries. All entries must be on a single line of text.

#### "Action" "ModelCriteria" Status //comment

**Action:** String describing the action associated with this element. See below for actions available.

**ModelCriteria:** This is the name of a ModelFilter or ModelCondition under which this action should be performed. This entry is optional. If it is not specified, then a blank (or no criteria) is assumed. If you want to enter a Status, then use must specify "" as the ModelCriteria.

**Status:** Three options

CHECK: perform action if ModelCriteria is true.

ALWAYS: perform action regardless of ModelCriteria.

NEVER: do not perform action.

This entry is optional. If it is not specified, then CHECK is assumed.

**Comment:** All text to the right of the comment symbol (//) will be saved with the CTGElement as a comment.

Possible Actions:

#### Calling of a name ContingencyBlock

CONTINGENCYBLOCK | name

Calls a ContingencyBlock and executes each of the actions in that block.

#### Transmission Line or Transformer outage or insertion

BRANCH | bus1# bus2# ckt | OPEN

| CLOSE

Takes branch out of service, or puts it in service. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus. Also, the entire sequence [bus1# bus2# ckt] may be replaced by the label of the branch.

#### Series Capacitor Bypass or In Service

SERIESCAP | bus1# bus2# ckt | BYPASS

| INSERVICE

Bypasses a series capacitor, or puts it in service. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus. Also, the entire sequence [bus1# bus2# ckt] may be replaced by the label of the branch.

#### DC Transmission Line outage or insertion

DCLine | bus1# bus2# ckt | OPEN  
| CLOSE

Takes DC Line out of service, or puts it in service. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus. Also, the entire sequence [bus1# bus2# ckt] may be replaced by the label of the dc transmission line.

DC Line set a specific value

DCLINE | bus1# bus2# ckt | SET\_P\_TO | value | MW  
| | CHANGE\_P\_BY | | PERCENT  
| | SET\_I\_TO | | AMPS  
| | CHANGE\_I\_BY

Use to set the DC line setpoint to a particular value. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus. Also, the entire sequence [bus1# bus2# ckt] may be replaced by the label of the dc transmission line.

Phase Shifter set a specific value

PHASESHIFTER | bus1# bus2# ckt | SET\_P\_TO | value | MW  
| | CHANGE\_P\_BY | | PERCENT

Use to set the phase shift angle to a particular value. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus. Also, the entire sequence [bus1# bus2# ckt] may be replaced by the label of the branch.

3-Winding Transformer outage or insertion

3WXFORMER | bus1# bus2# bus3# ckt | OPEN  
| CLOSE

Takes all three windings of a 3-winding transformer out of service, or puts them in service. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus. Also, the entire sequence [bus1# bus2# bus3# ckt] may be replaced by the label of the three winding transformer.

Interface outage or insertion

INTERFACE | name | OPEN  
| CLOSE

Takes all monitored branches in the interface out of service, or puts them all in service. Note: the [name] may be replaced by the label of the interface.

Bus outage causes all lines connected to the bus to be outage

BUS | bus# | OPEN

Takes all branches connected to the bus out of service. Also outages all generation, load, or shunts attached to the bus. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus.

#### Generator, Load, or Switched Shunt outage or insertion

GEN | bus# id | OPEN

LOAD | bus# id | CLOSE

SHUNT | bus#

INJECTIONGROUP | name

Takes a generator, load, or shunt out of service, or puts it in service. If specifying an injection group, the status of all devices in the injection group will be changed. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus. Also, the sequence [bus1# ckt] or [name] may be replaced by the label of the device.

#### Generator, Load or Switched Shunt movement to another bus

GEN | bus1# | MOVE\_PQ\_TO | bus2# | value | MW

LOAD | | MOVE\_P\_TO | | MVAR

SHUNT | | MOVE\_Q\_TO | PERCENT

Use to move generation, load or shunt at a bus1 over to bus2. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus.

#### Generator, Load or Switched Shunt set or change a specific value

GEN | bus# | SET\_P\_TO | value | MW

LOAD | | SET\_Q\_TO | | MVAR

SHUNT | | SET\_PQ\_TO | PERCENT

| SET\_VOLT\_TO |

| CHANGE\_P\_BY |

| CHANGE\_Q\_BY |

| CHANGE\_PQ\_BY |

| CHANGE\_VOLT\_BY |

Use to set the generation, load, or shunt at a bus to a particular value. Also can use to change by a specified amount. The voltage setpoints only apply to SHUNTS and GENs. Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in single quotes which represents the label of the bus.

#### Injection Group set a specific value

```
INJECTIONGROUP | name | SET_P_TO | value | MW
                | PERCENT
```

Use to set the MW generation/load in an injection group to a particular value.

#### Injection Group change specific value

```
INJECTIONGROUP | name | CHANGE_P_TO | value | MW
                | PERCENT
                | MWMERITORDER
                | PERCENTMERITORDER
```

Use to change the MW generation/load in an injection group by a particular value. Note that MW and PERCENT OPTIONS will change each point in the injection group by a value in proportion to the participations factors of the group. MWMERITORDER and PERCENTMERITORDER will modify points in the injection group by closing or opening points in the group in order of descending participation factors until the total change greater than or equal to the requested change has been achieved. Note: the [name] may be replaced by the label of the injection group.

#### Execute a Power Flow Solution

```
SOLVEPOWERFLOW
```

Include this action to cause the solution of the contingency to be split into pieces. Actions that are listed before each SOLVEPOWERFLOW call will be performed as a group.

#### Make-Up Power Compensation.

Only valid immediately following a SET, CHANGE, OPEN or CLOSE action on a Generator, Shunt or Load. This describes how the change in MW or MVAR are picked up by buses throughout the system. The values specify participation factors

```
COMPENSATION
```

```
bus#1 value1
```

```
bus#2 value2
```

```
...
```

```
END
```

Example:

```
<SUBDATA CTGElement>
```

```
// just some comments
```

```
// action
```

	Model Criteria	Status	comment
"BRANCH 40821 40869 1 OPEN"	""	ALWAYS	//Raver - Paul 500 kV
"GEN 45041 1 OPEN"	""	ALWAYS	//Trip Unit #2
"BRANCH 42702 42727 1 OPEN"	"Line X Limited"	CHECK	//Open Fern Hill
"GEN 40221 1 OPEN"	"Interface L1"	CHECK	//Drop ~600 MW
"GEN 40227 1 OPEN"	"Interface	CHECK	//Drop ~1200

"GEN 40221 1 OPEN"	L2"		MW
	"Interface	CHECK	//Drop ~600
	L3"		MW
"GEN 40227 1 OPEN"	"Interface	CHECK	//Drop ~1200
	L3+"		MW

&lt;/SUBDATA&gt;

Note: ContingencyElement object types can also be created inside their own DATA section as well. One of the key fields of the object is then the name of the contingency to which the ContingencyElement belongs.

### LimitViol

A LimitViol is used to describe the results of a contingency analysis run. Each Limit Violation lists eight values:

#### **ViolType ViolElement Limit ViolValue PTDF PTDF InitialValue Reason**

**ViolType:** One of five values describing the type of violation

BAMP: branch amp limit violation.

BMVA: branch MVA limit violation.

VLOW: bus low voltage limit violation.

VHIGH: bus high voltage limit violation.

INTER: interface MW limit violation.

**ViolElement:** This field depends on the ViolType.

for VLOW, VHIGH: "bus1#" or "busname\_buskv" or "buslabel"

for INTER: "interfacename" or "interfacelabel"

for BAMP, BMVA: "bus1# bus2# ckt violationbus# MWFlowDirection"

violationbus# is the bus number for the end of the branch which is violated

MWFlowDirection is the direction of the MW flow on the line. Potential values are "FROMTO" or "TOFROM"

Note: each bus# may be replaced with the name underscore nominal kV string enclosed in single quotes, or bus# values may be replaced by a string enclosed in single quotes representing the label of the bus. Also, the entire sequence [bus1# bus2# ckt] may be replaced by the label of the branch.

**Limit:** This is the numerical limit which was violated.

**ViolValue:** This is the numerical value of the violation.

**PTDF:** This field is optional. It only makes sense for interface or branch violations. It stores a sensitivity of the flow on the violating element during in the base case with respect to a transfer direction This must be calculated using the Contingency Analysis Other Actions related to Sensitivities.

**OTDF:** This field is optional. It only makes sense for interface or branch violations. It stores a sensitivity of the flow on the violating element during this contingency with respect to a transfer direction. This must be calculated using the Contingency Analysis Other Actions related to Sensitivities.

**InitialValue:** This stores a number. This stores the base case value for the element which is being violated. This is used to compare against when looking at change violations.

**Reason:** This will say whether this was a pure violation or is being reported as a violation because the change from the base case is higher than a specified threshold.

LIMIT - means this is a violation of a line/interface/bus limit

CHANGE - means this is being reported as a limit because the change from the initial value is higher than allowed

Example:

```
<SUBDATA LimitViol>  
BAMP "1 3 1 1 FROMTO" 271.94031 398.48096 10.0 15.01 // Note OTDF/PTDF  
INTER "Right-Top" 45.00000 85.84451 // Note OTDF/PTDF not included  
</SUBDATA>
```

Note: ViolationCTG object types can also be directly created inside their own DATA section as well. One of the key fields of the object is then the name of the contingency to which the ViolationCTG belongs.

### **Sim\_Solution\_Options**

These describe the power flow solution options which should be used under this particular contingency. The format of the subdata section is two lines of text. The first line is a list of the fieldtypes for Sim\_Solution\_Options which should be changed. The second line is a list of the values. Note that in general, power flow solution options are stored at three different locations in contingency analysis. When implementing a contingency, Simulator gives precedence to these three locations in the following order:

1. Contingency Record Options (stored with the particular contingency)
2. Contingency Tool Options (stored with CTG\_Options)
3. The global solution options

### **WhatOccurredDuringContingency**

Each line of this subdata section is part of a text description of what actually ended up being implemented for this contingency. This will list which actions were executed and which actions ended up being skipped because of their model criteria. Each line of the subdata section must be enclosed in quotes.



---

**Data CTG Options**

---

**Sim\_Solution\_Options**

These describe the power flow solution options which should be used under this particular contingency. The format of the subdata section is two lines of text. The first line is a list of the fieldtypes for Sim\_Solution\_Options which should be changed. The second line is a list of the values. Note that in general, power flow solution options are stored at three different locations in contingency analysis. When implementing a contingency, Simulator gives precedence to these three locations in the following order:

- Contingency Record Options (stored with the particular contingency)
- Contingency Tool Options (stored with CTG\_Options)
- The global solution options

---

## Data CTGElementBlock

---

### CTGElement

This format is the same as for the Contingency objecttype, however, you can not call a ContingencyBlock from within a contingencyblock.

### CTGElementAppend

When a subdata section is defined as **CTGElementAppend** rather than **CTGElement**, the actions of this subdata section will be appended to the contingency actions, instead of replacing them. This format is the same as for the Contingency objecttype, however, you can not call a ContingencyBlock from within a contingencyblock.

Note: CTGElementBlockElement object types can also be directly created inside their own DATA section as well. One of the key fields of the object is then the name of the contingency block to which the CTGElementBlockElement belongs.

---

**Data CustomCaseInfo**

---

**ColumnInfo**

Each line of this SUBDATA section can be used for specifying the column width of particular columns of the respective Custom Case Information Sheet. The line contains two values – the column and then a column width. This is shown in the following example.

**Example:**

```
<SUBDATA ColumnInfo>  
"SheetCol" 133  
"SheetCol:1" 150  
"SheetCol:2" 50  
</SUBDATA>
```

---

## Data CustomColors

---

### CustomColors

These describe the customized colors used in Simulator, which are specified by the user. A custom color is an integer describing a color. Each custom color is written on a single line of text and is an integer between 0 and 16,777,216. The value is determined by taking the red, green, and blue components of the color and assigning them a value between 0 and 255. The color is then equal to  $\text{red} + 256 * \text{green} + 256 * 256 * \text{blue}$ . Each line contains only one integer that corresponds to the color specified.

#### Example:

```
<SUBDATA CustomColors>
9823301
8613240
</SUBDATA>
```

---

**Data DataGrid**


---

**ColumnInfo**

Contains a description of the columns which are shown in the respective data grid. Each line of text contains four fields: **VariableName**, **ColumnWidth**, **TotalDigits**, **DecimalPoints**

**VariableName:** Contains the variable which is shown in this column.

**ColumnWidth:** The column width.

**TotalDigits:** The total digits displayed for numerical values.

**DecimalPoints:** The decimal points shown for numerical values.

**Example:**

```
DATA (DataGrid, [DataGridName])
```

```
{
  BUS
  <SUBDATA COLUMNINFO>
  BusNomVolt 100 8 2
  AreaNum 50 8 2
  ZoneNum 50 8 2
  </SUBDATA>
```

```
  BRANCHRUN
  <subdata COLUMNINFO >
  BusNomVolt: 0 100 8 2
  BusNomVolt: 1 100 8 2
  LineMW: 0 100 9 3
  </SUBDATA>
}
```

---

## Data DynamicFormatting

---

### DynamicFormattingContextObject

This subdata section contains a list of the display object types which are chosen to be selected. Each line of the section consists of the following:

**DisplayObjectType:** The object type of the display object. These are generally the same as the values seen in the subdata section SelectByCriteriaSetType of SelectByCriteriaSet object types. The only exception is the string *CaseInfo*, which is used for formatting applying to the case information displays.

**(WhichFields)** : For display objects that can reference different fields, this sets which of those fields it should select (e.g. select only Bus Name Fields). The value may be either ALL or SPECIFIED.

**(ListOfFields)** : If WhichFields is set to SPECIFIED, then a delimited list of fields follows.

#### Example:

```
<SUBDATA DynamicFormattingContextObject>
// Note: CaseInfo applies to case information displays
CaseInfo "SPECIFIED" BusName
DisplayAreaField "ALL"
DisplayBus
DisplayBusField "SPECIFIED" BusName BusPUVolt BusNum
DisplayCircuitBreaker
DisplaySubstation
DisplaySubstationField "SPECIFIED" SubName SubNum BusNomVolt BGLoadMVR
DisplayTransmissionLine
DisplayTransmissionLineField "ALL"
</SUBDATA>
```

### LineThicknessLookupMap

### LineColorLookupMap

### FillColorLookupMap

### FontColorLookupMap

### FontSizeLookupMap

### BlinkColorLookupMap

### XoutColorLookupMap

### FlowColorLookupMap

### SecondaryFlowColorLookupMap

The values of the lookup table for the characteristics that can be modified by the dynamic formatting tool. The first line contains the two following fields:

**fieldname:** It is the field that the lookup table is going to look for.

**usediscrete:** Set to YES or NO. If set to YES, the characteristic values will be discrete, meaning that the characteristic value will correspond exactly to the one specified in the

table. If set to NO, the characteristic values will be continuous, which means the characteristic value will be an interpolation of the high and low closest values specified in the table.

The following lines contain two fields:

**fieldvalue:** The value for the field.

**characteristicvalue:** The corresponding characteristic value for such field value.

**Example:**

```
<SUBDATA LineColorLookupMap>  
// FieldName UseDiscrete  
BusPUVolt YES  
// FieldValue Color  
1.02 16711808  
1.05 8454143  
1.1 16744703  
</SUBDATA>
```

---

## Data Filter

---

### Condition

Conditions store the conditions of the filter. Each condition is described by one line of text which can contain up to five fields.

#### **Variablename condition "value" ("othervalue") (fieldopt)**

**Variablename:** It is one of the fields for the object\_type specified. It may optional be followed by a colon and a non-negative integer. If not specified, 0 is assumed.

Example: on a LINE, 0 = from bus, 1 = to bus

Thus: LineMW: 0 = the MW flow leaving the from bus

LineMW: 1 = the MW flow leaving the to bus

**Condition:** Possible Values Alternate1 Alternate2 Requirements

between >< requires other value

notbetween ~>< requires other value

equal ==

notequal <> ~=

greaterthan >

lessthan <

greaterthanorequal >=

lessthanorequal <=

contains

notcontains

startswith

notstartswith

inrange

notinrange

meets

notmeets

isblank

notisblank

**value:** The value to compare to.

For fields associated with strings, this must be a string

For fields associated with real numbers, this must be a number.

For fields associated with integers, this is normally an integer, except when the Condition is "inrange" or "notinrange". In this case, value is a comma/dash separated number string.

**(othervalue):** If required, the other value to compare to.

**(FieldOpt):** Optional integer value with following meanings.

0 - strings are case insensitive, use number fields directly

(0 is the default value if not otherwise specified)



1 - strings are case sensitive, take ABS of field values

**Example:**

```
DATA (FILTER, [objecttype, filtername, filterlogic, filterpre])
{
  BUS "a bus filter" "AND" "no"
  <SUBDATA CONDITION>
  BusNomVolt > 100
  AreaNum inrange "1-5 , 7 , 90-95"
  ZoneNum between
  </SUBDATA>

  BRANCH "a branch filter" "OR" "no"
  <subdata CONDITION>
  BusNomVolt: 0 > 100 // Note location 0 means from bus
  BusNomVolt: 1 > 100 // Note location 1 means to bus
  LineMW: 0 > 100 1 // Note, final field 1 denotes absolute value
  </SUBDATA>
}
```

---

## Data Gen

---

### BidCurve

BidCurve subdata is used to define a piece-wise linear cost curve (or a bid curve). Each bid point consists of two real numbers on a single line of text: a MW output and then the respective bid (or marginal cost).

#### Example:

```
<SUBDATA BidCurve>
// MW Price[$/MWhr]
100.00 10.6
200.00 12.4
400.00 15.7
500.00 16.0
</SUBDATA>
```

### ReactiveCapability

Reactive Capability subdata is used to the reactive capability curve of the generator. Each line of text consists of three real numbers: a MW output, and then the respective Minimum MVAR and Maximum MVAR output.

#### Example:

```
<SUBDATA ReactiveCapability>
// MW MinMVAR MaxMVAR
100.00 -60.00 60.00
200.00 -50.00 50.00
400.00 -30.00 20.00
500.00 -5.00 2.00
</SUBDATA>
```

Note: ReactiveCapability object types can also be directly created inside their own DATA section as well. Two of the key fields of the object are then the bus number and generator ID of the generator to which the ReactiveCapability point belongs.

---

**Data GeoDataViewStyle**


---

**TotalAreaValueMap**

This subdata section is used to define the lookup table for determining the total area size of geographic data view objects based on the value of a selected field. Two values are entered for each mapping:

**FieldValue:** Value of the field selected for the Total Area attribute.

**TotalArea:** The total area size of the object.

**Example:**

```
<SUBDATA TotalAreaValueMap>
// FieldValue TotalArea
1.000 0
4.000 23
7.000 46
</SUBDATA>
```

**RotationRateValueMap**

This subdata section is used to define the lookup table for determining the rotation rate of geographic data view objects based on the value of a selected field. Two values are entered for each mapping:

**FieldValue:** Value of the field selected for the Rotation Rate attribute.

**RotationRate:** The rotation rate of the object. Entered in Hz.

**Example:**

```
<SUBDATA RotationRateValueMap>
// FieldValue RotationRate
1.000 0.00
4.000 0.10
7.000 0.20
</SUBDATA>
```

**RotationAngleValueMap**

This subdata section is used to define the lookup table for determining the rotation angle of geographic data view objects based on the value of a selected field. Two values are entered for each mapping:

**FieldValue:** Value of the field selected for the Rotation Angle attribute.

**RotationAngle:** The rotation angle of the object. Entered in degrees.

**Example:**

```
<SUBDATA RotationAngleValueMap>
```

```
// FieldValue RotationAngle  
1.000 -90.0  
4.000 0.0  
7.000 90.0  
</SUBDATA>
```

### **LineThicknessValueMap**

This subdata section is used to define the lookup table for determining the thickness of the border line around geographic data view objects based on the value of a selected field. Two values are entered for each mapping:

**FieldValue:** Value of the field selected for the Line Thickness attribute.

**LineThickness:** The line thickness of the border line around the object. This should be an integer value.

#### **Example:**

```
<SUBDATA LineThicknessValueMap>  
// FieldValue LineThickness  
1.000 1  
4.000 2  
7.000 3  
</SUBDATA>
```

---

**Data GlobalContingencyActions**

---

**CTGElementAppend**

This format is the same as for the Contingency objecttype

**CTGElement**

This format is the same as for the Contingency objecttype

Note: GlobalContingencyActionsElement object types can also be directly created inside their own DATA section as well.

---

## Data HintDefValues

---

### HintObject

Stores the values for the custom hints. Each line has one value:

**FieldDescription:** This is a string enclosed in double quotes. The string itself is delimited by the @ character. The string contains five values:

**Name of Field:** The name of the field. Special fields that appear on dialog by default have special names. Otherwise these are the same as the fieldnames of the AUX file format (for the "other fields" feature on the dialogs).

**Total Digit:** Number of total digits for a numeric field.

**Decimal Points:** Number of decimal points for a numeric field.

**Include Suffix:** Set to 0 for not including the suffix, and set to 1 to include it.

**Field Prefix:** The prefix text.

### Example:

```
<SUBDATA HintObject>
"BusPUVolt@4@1@1@PU Volt ="
"BusAngle@4@1@1@Angle ="
</SUBDATA>
```

---

**Data InjectionGroup**


---

**PartPoint**

A participation point is used to describe the contents of an injection group. Each participation point lists six values:

**PointType PointBusNum PointID PointParFac ParFacCalcType  
ParFacNotDynamic**

**PointType:** One of two values describing the type of violation:

GEN: a generator

LOAD: a load

INJECTIONGROUP: another injection group

**PointBusNum:** The bus number of the partpoint if the type is GEN or LOAD. Value will be blank for an injection group type. Note: bus# values may be replaced by a string enclosed in double quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. These values may also be replaced by a string enclosed in double quotes that represents the label of the bus or a string representing the label of the generator or load.

**PointID:** For a GEN or LOAD type, this is the id for the partpoint. For an INJECTIONGROUP type, this is the name or label of the injection group.

**PointParFac:** The participation factor for the point.

**ParFacCalcType:** How the participation point is calculated. There are several options depending on the PointType.

Generators: SPECIFIED, MAX GEN INC, MAX GEN DEC, or MAX GEN MW

Loads: SPECIFIED or LOAD MW

Injection Groups: SPECIFIED

**ParFacNotDynamic:** Should the participation factor be recalculated dynamically as the system changes.

**Example:**

```
<SUBDATA PartPoint
"GEN" 1 "1" 1.00 "SPECIFIED" "NO"
"GEN" 2 "1" 5.00 "SPECIFIED" "NO"
"GEN" 4 "1" 104.96 "MAX GEN INC" "NO"
"GEN" 6 "1" 50.32 "MAX GEN DEC" "YES"
"GEN" 7 "1" 600.00 "MAX GEN MW" "NO"
"LOAD" 2 "1" 5.00 "SPECIFIED" "NO"
"LOAD" 5 "1" 130.00 "LOAD MW" "NO"
"LOAD" 6 "1" 200.00 "LOAD MW" "YES"
</SUBDATA>
```

Note: PartPoint object types can also be directly created inside their own DATA section as well. One of the key fields of the PartPoint object is then the name of the injection group to which the participation point belongs.

---

**Data Interface**


---

**InterfaceElement**

An interface's subdata contains a list of the elements in the interface. Each line contains a text description of the interface element. Note that this text description must be encompassed by quotation marks. There are five kinds of elements allowed in an interface. Please note that the direction specified in the monitoring elements is important.

**"BRANCH num1 num2 ckt"** Monitor the MW flow on the branch starting from bus num1 going to bus num2 on the branch this branch. (order of bus numbers defines the direction)

**"AREA num1 num2"** Monitor the sum of the AC tie line MW flows from area num1 to area num2.

**"ZONE num1 num2"** Monitor the sum of the AC tie line MW flows from zone num1 to zone num2.

**"BRANCHOPEN num1 num2 ckt"** When monitoring the elements in this interface, monitor them under the contingency of opening this branch.

**"BRANCHCLOSE num1 num2 ckt"** When monitoring the elements in this interface, monitor them under the contingency of closing this branch.

**"DCLINE num1 num2 ckt"** Monitor the flow on a DC line.

**"INJECTIONGROUP 'name'"** Monitor the net injection from an injection group (generation contributes a positive injection, loads negative).

**"GEN num1 id"** Monitor the net injection from a generator (output is positive injection).

**"LOAD num1 id"** Monitor the net injection from a load (output is negative injection).

**"MSLINE num1 num2 ckt"** Monitor the MW flow on the multi-section line starting from bus num1 going to bus num2 with circuit ckt.

**"INTERFACE 'name'"** Monitor the MW flow on the interface given by name.

Note: bus# values may be replaced by a string enclosed in single quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus. Labels may also be used as follows:

- bus# values for all element types may be replaced by a string enclosed in single quotes where the string is the label of the bus
- for GEN or LOAD elements, the section *num1 id* may be replaced by the device's label
- for MSLINE, DCLINE, or BRANCH elements, the *num1 num2 ckt* section may be replaced by the device's label

For the interface element type "BRANCH num1 num2 ckt" and "DCLINE num1 num2 ckt", an optional field can also be written specifying whether the flow should be measured at the far end. This field is either YES or NO.

**Example:**

```
<SUBDATA InterfaceElement
```

```
"BRANCH 8 9 1" NO // monitor the flow from bus 8 to bus 9 on this branch
```

```
"BRANCH 12 33 1" YES // monitor the flow from bus 12 to bus 33 on branch
```



```
// measurefarend is set to true, therefore, we are
// monitoring the MW flow that arrives at bus 33

"AREA 2 1" // monitor tie line flow from area 2 to area 1
"ZONE 66 53" // monitor tie lines flows from zone 66 to zone 53
"BRANCHOPEN 5 6 1" // doe monitoring after branch opens
"BRANCHCLOSE 7 10 1" // doe monitoring after branch closes
</SUBDATA>
```

Note: InterfaceElement object types can also be directly created inside their own DATA section as well. One of the key fields of the InterfaceElement object is then the name of the interface to which the interface element belongs.

---

## Data LimitSet

---

### LimitCost

LimitCost records describe the piece-wise unenforceable constraint cost records for use by unenforceable line/interface limits in the OPF or SCOPF. Each row contains two values

**PercentLimit:** Percent of the transmission line limit.

**Cost:** Cost used at this line loading percentage value.

### Example:

```
<SUBDATA LimitCost>
//Percent Cost [$/MWhr]
100.00 50.00
105.00 100.00
110.00 500.00
</SUBDATA>
```

---

**Data Load**

---

**BidCurve**

BidCurve subdata is used to define a piece-wise linear benefit curve (or a bid curve). Each bid point consists of two real numbers on a single line of text: a MW output and then the respective bid (or marginal cost). These costs must be increasing for loads.

**Example:**

```
<SUBDATA BidCurve>  
// MW Price[$/MWhr]  
100.00 16.0  
200.00 15.7  
400.00 12.4  
500.00 10.6  
</SUBDATA>
```

---

**Data LPVariable**

---

**LPVariableCostSegment**

Stores the cost segments for the LP variables. Each line contains four values:

**Cost (Up):** Cost associated with increasing the LP variable.

**Minimum value:** Minimum limit of the LP variable.

**Maximum value:** Maximum limit of the LP variable.

**Artificial:** Whether the cost segment is artificial or not.

---

**Data ModelCondition**

---

**Condition**

ModelConditions are the combination of an object and a Filter. They are used to return when the particular object meets the filter specified. As a results, the subdata section here is identical to the Condition subdata section of a [Filter](#). See the description there.

---

**Data ModelExpression**


---

**LookupTable**

LookupTables are used inside Model Expressions sometimes. These lookup table represent either one or two dimensional tables. If the first string in the SUBDATA section is "x1x2", this signals that it is a two dimensional lookup table. From that point on it will read the first row as "x2" lookup points, and the first column in the remainder of the rows as the x1 lookup values.

**Example:**

```
DATA (MODELEXPRESSION, [CustomExpression,ObjectType,CustomExpressionStyle,
CustomExpressionString,WhoAmI,VariableName,WhoAmI:1,VariableName:1], AUXDEF)
{
// The following demonstrated a one dimensional lookup table
22.0000, "oneD", "Lookup", "",
"Gen<KEY1>1</KEY1><KEY2>1</KEY2>",
"Gen<KEY1>1</KEY1><KEY2>1</KEY2><VAR>GenMW</VAR>",
"", ""
',
<SUBDATA LookupTable>
// because it does not start with the string x1x2 this will
// represent a one dimensional lookup table
x1 value
0.000000 1.000000
11.000000 22.000000
111.000000 222.000000
</SUBDATA>

0.0000, "twod", "Lookup", "",
"Gen<KEY1>1</KEY1><KEY2>1</KEY2>",
"Gen<KEY1>1</KEY1><KEY2>1</KEY2><VAR>GenMW</VAR>",
"Gen<KEY1>6</KEY1><KEY2>1</KEY2>",
"Gen<KEY1>6</KEY1><KEY2>1</KEY2><VAR>GenMW</VAR>"
<SUBDATA LookupTable>
// because this starts with x1x2 this represent a two dimensional
// lookup table. The first column represents lookup values for x1.
// The first row represents lookup values for x2
x1x2 0.100000 0.300000 // these are lookup heading for x2
0.000000 1.000000 3.000000
11.000000 22.000000 33.000000
111.000000 222.000000 333.000000
</SUBDATA>
}
```

---

**Data ModelFilter**

---

**ModelCondition**

A Model Filter's subdata contains a list of each ModelCondition in the it. Because a list of Model Conditions is stored within Simulator, this subdata section only stores the name of each ModelCondition on each line.

**Example:**

```
<SUBDATA ModelCondition>  
"Name of First Model Condition"  
"Name of Second Model Condition"  
"Name of Third Model Condition"  
</SUBDATA>
```

---

## Data MTDCRecord

---

### MTDCBus

For this SUBDTA section, each DC Bus is described on a single line of text with exactly 8 fields specified.

**DCBusNum:** The number of the DC Bus. Note the DC bus numbers are independent of the AC bus numbers.

**DCBusName:** The name of the DC bus enclosed in quotes.

**ACTerminalBus:** The AC terminal to which this DC bus is connected (via a MTDCConverter). If the DC bus is not connected to any AC buses, then specify as zero. You may also specify this as a string enclosed in double quotes with the bus name followed by an underscore character, following by the nominal voltage of the bus.

**DCResistanceToGround:** The resistance of the DC bus to ground. Not used by Simulator.

**DCBusVoltage:** The DC bus voltage in kV.

**DCArea:** The area that this DC bus belongs to.

**DCZone:** The zone that this DC bus belongs to.

**DCOwner:** The owner that this DC bus belongs to.

Note: MTDCBus object types can also be directly created inside their own DATA section as well. One of the key fields of the object is then the number of the MTDCRecord to which the MTDCBus belongs.

### MTDCConverter

For this SUBDTA section, each AC/DC Converter is described by exactly 24 field which may be spread across several lines of text. Simulator will keep reading lines of text until it finds 24 fields. All text to the right of the 24<sup>th</sup> field (on the same line of text) will be ignored. The 24 fields are listed in the following order:

**BusNum:** AC terminal bus number.

**MTDCNBridges:** Number of bridges for the converter.

**MTDCConvEBas:** Converter AC base voltage.

**MTDCConvAngMxMn:** Converter firing angle.

**MTDCConvAngMxMn:1:** Converter firing angle max.

**MTDCConvAngMxMn:2:** Converter firing angle min.

**MTDCConvComm:** Converter commutating resistance.

**MTDCConvComm:1:** Converter commutating reactance.

**MTDCConvXFRat:** Converter transformer ratio.

**MTDCFixedACTap:** Fixed AC tap.

**MTDCConvTapVals:** Converter tap.

**MTDCConvTapVals:1:** Converter tap max.

**MTDCConvTapVals:2:** Converter tap min.

**MTDCConvTapVals:3:** Converter tap step size.

**MTDCConvSetVL:** Converter setpoint value (current or power).

**MTDCConvDCPF:** Converter DC participation factor.



**MTDCConvMarg:** Converter margin (power or current).

**MTDCConvType:** Converter type.

**MTDCMaxConvCurrent:** Converter Current Rating.

**MTDCConvStatus:** Converter Status.

**MTDCConvSchedVolt:** Converter scheduled DC voltage.

**MTDCConvIDC:** Converter DC current.

**MTDCConvPQ:** Converter real power.

**MTDCConvPQ:1:** Converter reactive power.

Note: MTDCConverter object types can also be directly created inside their own DATA section as well. One of the key fields of the object is then the number of the MTDCRecord to which the MTDCConverter belongs.

### MTDCTransmissionLine

For this SUBDTA section, each DC Transmission Line is described on a single line of text with exactly 5 fields specified:

**DCFromBusNum:** From DC Bus Number.

**DCToBusNum:** To DC Bus Number.

**CKTID:** The DC Circuit ID.

**Resistance:** Resistance of the DC Line in Ohms.

**Inductance:** Inductance of the DC Line in mHenries (Not used by Simulator).

### Example:

```
DATA (RECORD, [Num,Mode,ControlBus])
{
//-----
// The first Multi-Terminal DC Transmission Line Record
//-----
1 "Current" "SYLMAR3 (26098)"
<SUBDATA Bus>
//-----
// DC Bus data must appear on a single line of text
// The data consists of exactly 8 values
// DC Bus Num, DC Bus Name, AC Terminal Bus, DC Resistance to ground,
// DC Bus Voltage, DC Bus Area, DC Bus Zone, DC Bus Owner
//-----
3 "CEILO3P" 0 9999.00 497.92 40 404 1
4 "SYLMAR3P" 0 9999.00 439.02 26 404 1
7 "DC7" 41311 9999.00 497.93 40 404 1
8 "DC8" 41313 9999.00 497.94 40 404 1
9 "DC9" 26097 9999.00 439.01 26 404 1
10 "DC10" 26098 9999.00 439.00 26 404 1
</SUBDATA>
<SUBDATA Converter>
//-----
// convert subdata keeps reading lines of text until it has found
```

```

// values specified for 24 fields. This can span any number of lines
// any values to the right of the 24th field found will be ignored
// The next converter will continue on the next line.
//-----
41311 2 525.00 20.25 24.00 5.00 0.0000 16.3100
0.391048 1.050000 1.000000 1.225000 0.950000 0.012500
1100.0000 1650.0000 0.0000 "Rect" 1650.0000 "Closed"
497.931 1100.0000 547.7241 295.3274
41313 4 232.50 15.36 17.50 5.00 0.0000 7.5130
0.457634 1.008700 1.030000 1.150000 0.990000 0.010000
2000.0000 2160.0000 0.1550 "Rect" 2160.0000 "Closed"
497.940 2000.0000 995.8800 561.8186
26097 2 230.00 20.90 24.00 5.00 0.0000 16.3100
0.892609 1.000000 1.100000 1.225000 0.950000 0.012500
-1100.0000 1650.0000 "" "Inv" 1650.0000 "Closed"
439.009 1100.0000 -482.9099 274.5227
26098 4 232.00 17.51 20.00 5.00 0.0000 7.5130
0.458621 1.008700 1.100000 1.120000 0.960000 0.010000
439.0000 2160.0000 "" "Inv" 2160.0000 "Closed"
439.000 1999.9999 -878.0000 544.2775
</SUBDATA>
<SUBDATA TransmissionLine>
//-----
// DC Transmission Segment information appears on a single line of
// text. It consists of exactly 5 value
// From DCBus, To DCBus, Circuit ID, Line Resistance, Line Inductance
//-----
3 4 "1" 19.0000 1300.0000
7 3 "1" 0.0100 0.0000
8 3 "1" 0.0100 0.0000
9 4 "1" 0.0100 0.0000
10 4 "1" 0.0100 0.0000
</SUBDATA>

//-----
// A second Multi-Terminal DC Transmission Line Record
//-----
2 "Current" "SYLMAR4 (26100)"
<SUBDATA Bus>
5 "CELILO4P" 0 9999.00 497.92 40 404 1
6 "SYLMAR4P" 0 9999.00 439.02 26 404 1
11 "DC11" 41312 9999.00 497.93 40 404 1
12 "DC12" 41314 9999.00 497.94 40 404 1
13 "DC13" 26099 9999.00 439.01 26 404 1
14 "DC14" 26100 9999.00 439.00 26 404 1
</SUBDATA>
<SUBDATA Converter>
41312 2 525.00 20.26 24.00 5.00 0.0000 16.3100
0.391048 1.050000 1.000000 1.225000 0.950000 0.012500
1100.0000 1650.0000 0.0000 "Rect" 1650.0000 "Closed"
497.931 1100.0000 547.7241 295.3969

```

```

41314 4 232.50 15.45 17.50 5.00 0.0000 7.5130
0.457634 1.008700 1.030000 1.150000 0.990000 0.010000
2000.0000 2160.0000 0.1550 "Rect" 2160.0000 "Closed"
497.940 2000.0000 995.8800 562.9448
26099 2 230.00 20.90 24.00 5.00 0.0000 16.3100
0.892609 1.000000 1.100000 1.225000 0.950000 0.012500
-1100.0000 1650.0000 "" "Inv" 1650.0000 "Closed"
439.009 1100.0000 -482.9099 274.5227
26100 4 232.00 17.51 20.00 5.00 0.0000 7.5130
0.458621 1.008700 1.100000 1.120000 0.960000 0.010000
439.0000 2160.0000 "" "Inv" 2160.0000 "Closed"
439.000 1999.9999 -878.0000 544.2775
</SUBDATA>
<SUBDATA TransmissionLine>
5 6 "1" 19.0000 1300.0000
11 5 "1" 0.0100 0.0000
12 5 "1" 0.0100 0.0000
13 6 "1" 0.0100 0.0000
14 6 "1" 0.0100 0.0000
</SUBDATA>
}

```

Note: MTDCTransmissionLine object types can also be directly created inside their own DATA section as well. One of the key fields of the object is then the number of the MTDCRecord to which the MTDCTransmissionLine belongs.

---

**Data MultiSectionLine**


---

**Bus**

A multi section line's subdata contains a list of each dummy bus, starting with the one connected to the From Bus of the MultiSectionLine and proceeding in order to the bus connected to the To Bus of the Line. Note: bus# values may be replaced by a string enclosed in double quotes where the string is the name of the bus followed by an underscore character and then the nominal voltage of the bus, or the string may represent the label of the bus.

**Example:**

```
//-----
-

// The following describes a multi-section line that connects bus
// 2 - 1 - 5 - 6 - 3
//-----
--

DATA (MultiSectionLine, [BusNum, BusName, BusNum: 1, BusName: 1,
  LineCircuit, MSLineNSections, MSLineStatus] )
{
  2 "Two" 3 "Three" "&1" 2 "Closed"
  <SUBDATA Bus>
  1
  5
  6
  </SUBDATA>
}
```

---

**Data Nomogram**

---

**InterfaceElementA****InterfaceElementB**

InterfaceElementA values represent the interface elements for the first interface of the nomogram. InterfaceElementB values represent the interface elements for the second interface of the nomogram. The format of these SUBDATA sections is identical to the format of the InterfaceElement SUBDATA section of a normal Interface.

**NomogramBreakPoint**

This subdata section contains a list of the vertex points on the nomogram limit curve.

**Example:**

```
<SUBDATA NomogramBreakPoint>
// LimA LimB
-100 -20
-100 100
80 50
60 -10
</SUBDATA>
```

---

**Data Owner**


---

**Bus**

This subdata section contains a list of the buses which are owned by this owner. Each line of text contains the bus number. As an alternative to specifying the bus number, a string enclosed in double quotes may be used where the string represents the name of the bus followed by an underscore character and then the nominal voltage of the bus, or the string may represent the label of the bus.

**Example:**

```
<SUBDATA Bus>
1
35
65
</SUBDATA>
```

**Load**

This subdata section contains a list of the loads which are owned by this owner. Each line of text contains the bus number followed by the load id. As an alternative to specifying the bus number, a string enclosed in double quotes may be used where the string represents the name of the bus followed by an underscore character and then the nominal voltage of the bus, or the string may represent the label of the bus. Also, instead of specifying the bus and load id, the label of the load enclosed in double quotes may be used.

**Example:**

```
<SUBDATA Load>
5 1 // shows ownership of the load at bus 5 with id of 1
423 1
</SUBDATA>
```

**Gen**

This subdata section contains a list of the generators which are owned by this owner and the fraction of ownership. Each line of text contains the bus number, followed by the gen id, followed by an integer showing the fraction of ownership. As an alternative to specifying the bus number, a string enclosed in double quotes may be used where the string represents the name of the bus followed by an underscore character and then the nominal voltage of the bus, or the string may represent the label of the bus. Also, instead of specifying the bus and generator id, the label of the generator enclosed in double quotes may be used.

**Example:**

```
<SUBDATA Gen>
78 1 50 // shows 50% ownership of generator at bus 78 with id of 1
23 3 70
</SUBDATA>
```

**Branch**

This subdata section contains a list of the branches which are owned by this owner and the fraction of ownership. Each line of text contains the from bus number, followed by the to bus number, followed by the circuit id, followed by an integer showing the fraction of ownership. As an alternative to specifying the bus numbers, strings enclosed in double quotes may be used where the string represents the name of the bus followed by an underscore character and then the nominal voltage of the bus, or the string may represent the label of the bus. Also, instead of specifying the two numbers and a circuit id, the label of the branch enclosed in double quotes may be used.

**Example:**

```
<SUBDATA Branch>  
6 10 1 50 // shows 50% ownership of line from bus 6 to 10, circuit 1  
</SUBDATA>
```

---

## Data PieChartGaugeStyle

---

### ColorMap

This is a lookup table by percentage of scalar and color values. This lookup table will consist of consecutive lines of text with exactly three values:

**Percentage:** This is the percentage at which the following scalar and color should be applied.

**Scalar:** A scalar (multiplier) on the size of the pie chart/gauge.

**Color:** A color for the pie chart/gauge.

Example:

```
<SUBDATA ColorMap>
//Percentage Scalar Color
85.0000 1.5000 33023
100.0000 2.0000 255
</SUBDATA>
```



---

**Data PostPowerFlowActions**

---

**CTGElementAppend**

This format is the same as for the Contingency objecttype.

**CTGElement**

This format is the same as for the Contingency objecttype.

Note: PostPowerFlowActionsElement object types can also be directly created inside their own DATA sections as well.

---

**Data PWCaseInformation**

---

**PWCaseHeader**

This subdata section contains the Case Description in free-formatted text. Note: as it is read back into Simulator all spaces from the start of each line are removed.

---

**Data PWFormOptions**

---

**PieSizeColorOptions**

There can actually be several PieSizeColorOptions subdata sections for each PWFormOptions object. The first line of each subdata section, the first line of text consist of exactly four values

**ObjectName:** The objectname of the type of object these settings apply to. Will be either be BRANCH or INTERFACE.

**FieldName:** The fieldname for the pie charts that these settings apply to.

**UseDiscrete:** Set to YES to use a discrete mapping of colors and size scalars instead of interpolating for intermediate values.

**UseOtherSettings:** Set to YES to default these settings to the BRANCH MVA values for BRANCH object. This allows you to apply the same settings to all pie charts.

After this first line of text, if the UseOtherSettings Value is NO, then another line of text will contain exactly three values:

**ShowValue:** This is the percentage at which the value should be drawn on the pie chart.

**NormalSize:** This is the scalar size multiplier which should be used for pie charts below the lowest percentage specified in the lookup table.

**NormalColor:** This is the color which should be used for pie charts below the lowest percentage specified in the lookup table.

Finally the remainder of the subdata section will contain a lookup table by percentage of scalar and color values. This lookup table will consist of consecutive lines of text with exactly three values

**Percentage:** This is the percentage at which the follow scalar and color should be applied.

**Scalar:** A scalar (multiplier) on the size of the pie charts.

**Color:** A color for the pie charts.

---

**Data PWLPOPFGTGViol**

---

**OPFControlSense****OPFBusSenseP****OPFBusSenseQ**

This stores the control sensitivities for each contingency violation during OPF/SCOPF analysis. Each line contains one value:

**Sensitivity:** The value of the sensitivity with respect to each control in OPFControlSense or with respect to each bus in OPFBusSenseP and OPFBusSenseQ.

**Example:**

```
<SUBDATA OPFControlSense>
// Value
1.000441679
2.447185E-7
-1.1109307E-6
1.6427327E-7
0
</SUBDATA>
```

---

**Data PWLPTabRow**

---

**LPBasisMatrix**

This subdata section stores the basis matrix associated with the final LP OPF solution. Each line contains two values:

**Variable:** The basic variable.

**Value:** The sensitivity of the constraint to the basic variable.

**Example:**

```
<SUBDATA LPBasisMatrix>
// Var Value
1 1.00000
2 1.00000
5 1.00000
6 1.00000
</SUBDATA>
```

---

**Data PWPVResultListContainer**


---

**PWPVResultObject**

This subdata section contains the results of a particular PV Curve scenario (a particular contingency or the base case). The data consists of two general sections: the first three rows of text contain the "independent axis" of the PV Curve. The first row starts with the string INDNOM and is followed by a list of numbers representing the nominal shift, the second row starts with INDEXP and is followed by the export shift, and the third row starts with INDIMP and is followed by the import shift. Following After these rows is a list of all the tracked quantities. Each tracked quantity row consists of three parts which are separated by the strings ?f= and &v= . The first part of the string represents a description of the power system object being tracked, the second part represents the field name being tracked, and the third contains a list of all the values at the various shift levels.

**Example:**

```
<SUBDATA PWPVResultObject>
INDNOM 0.00 500.00 1000.00 1500.00 1750.00 1875.00 1975.00
INDEXP 0.00 500.00 1000.00 1500.00 1750.00 1875.00 1975.00
INDIMP 0.00 -417.23 -701.58 -890.58 -952.60 -975.35 -990.43
Bus '3'?f=BusPUVolt&v= 0.993 0.983 0.964 0.939 0.926 0.919 0.914
Bus '5'?f=BusPUVolt&v= 1.007 1.000 0.982 0.956 0.940 0.932 0.926
Gen '4' '1'?f=GenMVR&v= 19.99 245.27 523.62 831.13 986.84 1060.6 1118.7
Gen '6' '1'?f=GenMVR&v= -6.59 -120.84 -131.37 -39.53 48.35 103.8 154.5
</SUBDATA>
```

---

**Data QVCurve**


---

**QVPoints**

This subdata section contains a list of the QV Curve points calculated for the respect QVCurve. Each line consists of exactly four values:

**PerUnitVoltage:** The per unit voltage of the bus for a QV point.

**FictitiousMvar:** The amount of Mvars injected by the fictitious generator at this QV point.

**ShuntDeviceMvar:** The Mvars injected by any switched shunts at the bus.

**TotalMvar:** The total Mvars injected by switched shunts and the fictitious generator.

**Example:**

```
DATA (QVCURVE,
[BusNum,CaseName,qv_VQ0,qv_Q0,qv_Vmax,qv_QVmax,qv_VQmin,qv_Qmin,
qv_Vmin,qv_QVmin,Qinj_Vmax,Qinj_0,Qinj_min,Qinj_Vmin])
{
5 "BASECASE" 0.880 0.000 1.100 312.490 0.480 -221.072
0.180 -86.334 191.490 -77.373 -244.075 -89.562
<SUBDATA QVPoints>
// NOTE: This bus has a constant impedance
// switched shunt value of -100 Mvar at it.
1.1000, 312.4898, -121.0000, 191.4898
0.9800, 124.6619, -95.9656, 28.6963
0.7800, -96.6202, -60.7808, -157.4010
0.5800, -206.9895, -33.5960, -240.5855
0.3800, -207.4962, -14.4113, -221.9075
0.1800, -86.3336, -3.2284, -89.5620
</SUBDATA>
}
```

---

**Data QVCurve\_Options**

---

**Sim\_Solution\_Options**

This subdata section contains solution options that will be used when running QV Curves. See explanation under the CTG\_Options object type for more information.



---

## Data SelectByCriteriaSet

---

### SelectByCriteriaSetType

This subdata section contains a list of the display object types which are chosen to be selected. Each line of the section consists of the following:

**DisplayObjectType:** The object type of the display object.

**(FilterName):** A filter to apply to these types of objects. This field is optional, but must be given if either of the following fields are given.

**(WhichFields):** For display objects that can reference different fields, this sets which of those fields it should select (e.g. select only Bus Name Fields). The value may be either ALL or SPECIFIED.

**(ListOfFields):** If WhichFields is set to SPECIFIED, then a delimited list of fields follows.

#### Example:

```
<SUBDATA SelectByCriteriaSetType>
DisplayAreaField "" "ALL"
DisplayBus ""

DisplayBusField "Name of Bus Filter" "SPECIFIED" BusName BusPUVolt BusNum
DisplayCircuitBreaker ""
DisplaySubstation ""
DisplaySubstationField "" "SPECIFIED" SubName SubNum BusNomVolt BGLoadMVR
DisplayTransmissionLine ""
DisplayTransmissionLineField "" "ALL"
</SUBDATA>
```

### Area

This subdata section contains a list of areas which were chosen to be selected. Each line of the section consists of either the number or the name. When generated automatically by PowerWorld we also include the other identifier as a comment.

#### Example:

```
<SUBDATA Area>
18 // NEVADA
22 // SANDIEGO
30 // PG AND E
52 // AQUILA
</SUBDATA>
```

### Zone

This subdata section contains a list of zones which were chosen to be selected. Each line of the section consists of either the number or the name. When generated automatically by PowerWorld we also include the other identifier as a comment.

#### Example:

```
<SUBDATA Zone>  
680 // ID SOLUT  
682 // WY NE IN  
</SUBDATA>
```

### **ScreenLayer**

This subdata section contains a list of screen layers which were chosen to be selected. Each line of the section consists of either the name.

#### **Example:**

```
<SUBDATA ScreenLayer>  
"Border"  
"Transmission Line Objects"  
</SUBDATA>
```

---

**Data ShapeFileExportDescription**

---

This object uses the same subdata sections as **SelectByCriteriaSet**. The only distinction is that only buses and lines can be exported.

---

## Data StudyMWTransactions

---

### ImportExportBidCurve

This subdata section contains the piecewise linear transactions cost curves for areas involved in a MW transaction. Costs are only for areas that are not on OPF control. Curves must be monotonically increasing. Each line corresponds to a point in the cost curve, and it has two values:

**MW:** The MW value. Use negative values for imports (purchase) and positive values for exports (sales).

**Price:** The price in \$/MWh.

#### Example:

```
<SUBDATA ImportExportBidCurve>
//MW Price[$/MWh]
-20.00 5.00
-10.00 10.00
0.00 15.00
10.00 20.00
20.00 45.00
30.00 70.00
</SUBDATA>
```

---

**Data SuperArea**

---

**SuperAreaArea**

This subdata section contains a list of areas within each super area. Each line of text contains two values, the area number followed by a participation factor for the area that can be optionally used.

**Example:**

```
<SUBDATA SuperAreaArea>  
1 48.9  
2 34.2  
25 11.2  
</SUBDATA>
```

---

## Data TSSchedule

---

### SchedPointList

This section stores the schedule time points used in Time Step Simulation. Each line contains seven values:

**Date:** The date of the point.

**Hour:** The hour of the point.

**Pointtype:** An integer specifying the point type.

0 – Numeric

1 – Boolean (Yes/No, Closed/Open)

2 – Text

**Numeric Value:** The numeric value if point type is Numeric. Otherwise it is just zero.

**Boolean Value:** The boolean value if point type is Boolean. Otherwise it is just false.

**Text Value:** The text value if point type is Text. Otherwise it is just an empty string.

**Audiofilename:** The audio filename associated to the point. If none, it is just an empty string.

Example:

```
<SUBDATA SchedPointList>
//Date Hour PointType NValue BValue TValue AValue
5/8/2006 0 1.00 NO
5/8/2006 6:00:00 AM 0 1.10 NO
5/8/2006 12:00:00 PM 0 1.25 NO
</SUBDATA>
```

---

**Data View**

---

**ScreenLayer**

This is a list of screen layer names that are hidden in the current view. Each screen layer name is on a separate line of text.

**Example:**

```
<SUBDATA ScreenLayer>  
//These are hidden screen layers  
"pie layer"  
</SUBDATA>
```

---

**Data Section for Display Auxiliary File**


---

The Data Section of the script file format can contain numerous functions for modifying the data of a power system case. The DATA section begins with a left curly brace and ends with a right curly brace.

**DATA DataName (object\_type, [list\_of\_fields], file\_type\_specifier)**

```
{
data_list_1
.
.
.
data_list_n
}
```

Immediately following the DATA keyword, you may optionally include a DataName. Currently, this is not used for anything because there are no script commands that are supported with the *display auxiliary* files.

The first line of the DATA section is referred to as the [Data Argument List](#). The [Data Argument List](#) defines the type and format of the data contained in the following section enclosed in curly braces.

Inside of each DATA section, [data lists](#) are given according to the data definition given in the [Data Argument List](#).



---

## Data Argument List for Display Auxiliary File

---

The DATA argument list identifies what the information section contains. A left and right parenthesis "(" ")" mark the beginning and end of the argument list.

The **file\_type\_specifier** parameter distinguishes the information section as containing custom auxiliary data (as opposed to Simulator's native auxiliary formats), and indicates the format of the data. Currently, the parser recognizes two values for **file\_type\_specifier**:

(blank) or AUXDEF	Data fields are space delimited
CSV, AUXCSV, or CSVAUX	Data fields are comma delimited

The **object\_type** parameter identifies the type of object or data element the information section describes or models. For example, if **object\_type** equals DISPLAYBUS, then the data describes DISPLAY BUS objects. Below is a list of many of the object types recognized by Simulator:

CaseInformationMemo	DisplayLineFlowArrow	DisplayTransmissionLineField
ColorMap	DisplayLoad	DisplayZone
Contour	DisplayLoadField	DisplayZoneField
CustomColors	DisplayMultiSectionLine	DisplayZonePie
DisplayArea	DisplayMultiSectionLineField	DocumentLink
DisplayAreaField	DisplayMultiSectionLinePie	DynamicFormatting
DisplayAreaPie	DisplayOwner	Ellipse
DisplayBranchGauge	DisplayOwnerField	Filter
DisplayBranchPie	DisplayOwnerPie	GeoDataViewStyle
DisplayBus	DisplaySeriesCapacitor	GeographyDisplayOptions
DisplayBusField	DisplaySeriesCapacitorField	Group
DisplayBusPie	DisplayShunt	Line
DisplayCircuitBreaker	DisplayShuntField	OnlineField
DisplayDCTransmissionLine	DisplayShuntPie	OnlineLink
DisplayDCTransmissionLineField	DisplaySubstation	Picture
DisplayGen	DisplaySubstationField	PieChartGaugeStyle
DisplayGenericModelField	DisplaySubstationPie	PWFormOptions
DisplayGenField	DisplaySuperArea	Rectangle
DisplayGenPie	DisplaySuperAreaField	Screenlayer
DisplayInjectionGroup	DisplaySuperAreaPie	SelectByCriteriaSet
DisplayInjectionGroupPie	DisplayTransformer	Text
DisplayInterface	DisplayTransformerField	TextBox
DisplayInterfaceField	DisplayTransformerPie	View
DisplayInterfacePie	DisplayTransmissionLine	

The list of object types Simulator's auxiliary file parser can recognize will grow as new applications for the technology are found. Within Simulator, you will always be able to obtain a list of the available **object\_types** by going to the **Auxiliary Files** ribbon group on the [Window](#) ribbon tab and choosing **Export Display Object Fields**.

The **list\_of\_fields** parameter lists the types of values the ensuing records in the data section contain. The order in which the fields are listed in **list\_of\_fields** dictates the order in which the fields will be read from the file. Simulator currently recognizes over 2,000 different field types, each identified by a specific field name. Because the available fields for an object may grow as new applications are developed, you will always be able to obtain a list of the available **object\_types** by going to the **Auxiliary Files** ribbon group on the [Window](#) ribbon tab and choosing **Export Display Object Fields**. Certainly, only a subset of these fields would be found in a typical custom auxiliary file. In crafting applications to export custom auxiliary files, developers need concern themselves only the fields they need to communicate between their applications and Simulator. A few points of interest regarding the **list\_of\_fields** are:

- The **list\_of\_fields** may take up several lines of the text file.
- The **list\_of\_fields** should be encompassed by braces [ ].
- When encountering the PowerWorld comment string `//` in one of these lines of the text file, all text to the right of this is ignored.
- Blank lines, or lines whose first characters are `//` will be ignored as comments.
- Field names must be separated by commas.

#### Example DATA Argument List

```
DATA (DISPLAYBUS, [BusNom, SOAuxiliaryID, // comment here
SOX, SOY, SOTickness, SOColor, SOUseFillColor, SOFillColor,
// comments allowed here too

// note that blank rows are ignored
SSize, SOWidth, SOOrientation, SOLevel, SOImmobile,
SLName, SOSStyle, SODashed, // more comments
SOBelongsToGroup])
```

One general note regarding the field names however. Some field names may be augmented with a field location. One example of this is the field `BusNum` when identifying branches. Bus numbers must be used to identify the from and the to end of branches. To keep the number of fields from becoming too large, the same field name is used for both of these values. The from bus number is written as `BusNum:0` and the to bus number is written as `BusNum:1`. Note that the `:0` may be left off of field names.

---

## Data Key Fields for Display Auxiliary File

---

Simulator uses certain fields to identify the specific object being described. These fields are called key fields. For example, the key field for DISPLAYBUS objects is BusNum, because a bus can be identified uniquely by its number. The key fields for DISPLAYGEN objects are BusNum and GenID. To properly identify each object, the object's key fields must be present. They can appear in any order in the **list\_of\_fields** (i.e. they need not be the first fields listed in **list\_of\_fields**). As long as the key fields are present, Simulator can identify the specific object.

Display objects have an additional key field used for identification because multiple objects can be present on the same one-line diagram that represent the same power system element. This extra key field is SOAuxiliaryID. This is a field that is unique for each type of display object and other key field combination. If there are two display buses that represent bus one in the power system, the SOAuxiliaryID field will be different for both. Simulator will automatically create unique identifiers when these objects are created graphically. They can also be user specified but are forced to be unique. This field does not need to be present when reading in a display auxiliary file, but if it is missing, Simulator assumes that the ID is "1". This field is the only key field identifier for objects that do not link to power system elements such as background lines and pictures, and therefore, should always be included when reading in these objects or the expected results may not be achieved.

By going to the **Auxiliary Files** ribbon group on the [Window](#) ribbon tab and choosing **Export Display Object Fields** you will obtain a list of fields available for each object type. In this output, the key fields will appear with asterisks \*.

---

## Data List for Display Auxiliary File

---

After the data argument list is completed, the Data list is given. The data section lists the values of the fields for each object in the order specified in **list\_of\_fields**. The data section begins with a left curly brace and ends with a right curly brace. A few points of interest regarding the **value\_list**:

- The **value\_list** may take up several lines of the text file.
- Each new data object must start on its own line of text.
- When encountering the PowerWorld comment string `'//'` in one of these lines of the text file, all text to the right of this is ignored.
- Blank lines, or lines whose first characters are `'//'` will be ignored as comments.
- Remember that the right curly brace must appear on its own line at the end of the **data\_list**.
- If the **file\_type\_specifier** is CSV, the values should be separated by commas. Otherwise, separate the field names using spaces.
- Strings can be enclosed in double quotes, but this is not required. You should however always enclose strings that contain spaces (or commas) in quotes. Otherwise, strings containing commas would cause errors for comma-delimited files, and spaces would cause errors for space-delimited formatted files.

---

## Special Data Sections

---

There are several object types that should be noted here because they can impact the reading of an entire display auxiliary file, overall look of the resulting one-line diagram, or require special input to properly import/export the object.

### GeographyDisplayOptions

Most objects supported in the display auxiliary file have coordinates that can be specified in the appropriate data sections. What these coordinates specify can be controlled by the GEOGRAPHYDISPLAYOPTIONS object. This object has only two fields available: MapProjection and ShowLonLat. There are three possible settings for MapProjection: "x,y", "Simple Conic", and "Mercator". The choice of projection will determine how the x,y values for display objects are interpreted. ShowLonLat can be either "YES" or "NO". If ShowLonLat is "YES", the setting specified for the MapProjection will be the longitude,latitude projection used when reading/writing the object x,y values. If ShowLonLat is "NO", the x,y values will always be interpreted as x,y regardless of the MapProjection setting. This object should be placed in the display auxiliary file before any other objects containing coordinates are read. If this object is not included in the auxiliary file, the coordinates will be interpreted based on the current settings of map projection and whether or not coordinates are showing longitude,latitude.

### Picture

PICTURE objects represent background images that cannot be stored in a text file format. To properly include a PICTURE object in a display auxiliary file, the file containing the image must be saved and read along with the auxiliary file. The FileName field indicates the name and location of the image file. If the image file cannot be found when reading in a display auxiliary file and attempting to create a new object, no PICTURE object will be created. If attempting to update an existing object and the image file cannot be found, the object will not be updated with a new image, but the FileName field will be updated with the specified file name.

### PWFormOptions

One-line display options that affect the current display settings can be changed by using the PWFORMOPTIONS object. Usually, this object specifies named sets of options that can be selected and used to change the various one-line display options through the GUI. By including a specially named object, the current options can be changed through a display auxiliary file. PWFORMOPTIONS are named using the OOName field. Setting this field to "THESE\_OPTIONS\_ARE\_APPLIED\_TO\_THE\_CURRENT\_DISPLAY" will apply the specified set of options to the current one-line when the file is read. When saving the entire one-line to a display auxiliary file, a PWFORMOPTIONS object with this name is added to the file by default.

### View

Different views can be specified in the display auxiliary file using the VIEW object. Usually, this object is used to specify named sets of options used to select and change the view through the GUI. By including a specially named object, the current view can be changed through a display auxiliary file. VIEW objects are named using the ViewName field. Setting this field to "THIS\_VIEW\_IS\_APPLIED\_TO\_THE\_CURRENT\_DISPLAY" will apply the specified set of view options to the current one-line when the file is read. When saving the entire one-line to a display auxiliary file, a VIEW object with this name is added to the file by default.

---

**Data Subdata Sections for Display Auxiliary File**


---

The format described in the [Data Section](#) works well for most kinds of data in Simulator. It does not work as well however for data that stores a list of objects. For example, a contingency stores some information about itself (such as its name), and then a list of contingency elements, and possible a list of limit violations as well. For data such as this, Simulator allows `<SubData>`, `</SubData>` tags that store lists of information about a particular object. This formatting looks like the following:

**DATA (object\_type, [list\_of\_fields], file\_type\_specifier)**

```
{
value_list_1
  <SUBDATA subobject_type1>
    precise format describing an object_type1
    precise format describing an object_type1
    .
    .
    .
  </SUBDATA>
  <SUBDATA subobject_type2>
    precise format describing an object_type2
    precise format describing an object_type2
    .
    .
    .
  </SUBDATA>
value_list_2
  .
  .
  .
value_list_n
}
```

Note that the information contained inside the `<SubData>`, `</SubData>` tags may not be flexibly defined. It must be written in a precisely defined order that will be documented for each SubData type.

---

## Data Common Sections

---

### **ColorMap**

Same format as in *data auxiliary* files.

### **CustomColors**

Same format as in *data auxiliary* files.

### **DynamicFormatting**

Same format as in *data auxiliary* files.

### **Filter**

Same format as in *data auxiliary* files.

### **GeoDataViewStyle**

Same format as in *data auxiliary* files.

### **PWFormOptions**

Same format as in *data auxiliary* files.

### **SelectByCriteriaSet**

Same format as in *data auxiliary* files.

---

**Data DisplayLine**


---

**DisplayDCTramisssionLine****DisplayInterface****DisplayMultiSectionLine****DisplaySeriesCapacitor****DisplayTransformer****DisplayTransmissionLine****Line**

This is a list of points defining the graphical line used to represent the object. Each set of coordinates can be enclosed in square brackets, [ ], or the brackets can be eliminated. The brackets will be included when Simulator generates an auxiliary file. The individual coordinates are separated by the specified delimiter, either a space or a comma, and if the brackets are included, the same delimiter should be used to separate sets of coordinates. The list of points is in a somewhat free form and sets of coordinates can span multiple lines. Each point should either be in x,y coordinates or longitude,latitude coordinates. Which coordinates should be used depends on the current option settings for map projection and whether or not coordinates should be shown in longitude,latitude. If the display auxiliary file is automatically generated by Simulator, a comment will be included in the subdata section indicating the coordinate system in use during file creation.

**Example using brackets and a comma delimiter:**

```
<SUBDATA Line>
//Coordinates are x,y
[14.00000000, 63.00000000], [14.00000000, 60.00000000],
[20.00000000, 45.00000000], [20.00000000, 42.00000000]
</SUBDATA>
```

**Example with no brackets and a space delimiter:**

```
<SUBDATA Line>
//Coordinates are x,y
14.00000000 63.00000000 14.00000000 60.00000000
20.00000000 45.00000000 20.00000000 42.00000000
</SUBDATA>
```



---

**Data PieChartGaugeStyle**

---

**ColorMap**

This is a lookup table by percentage of scalar and color values. This lookup table will consist of consecutive lines of text with exactly three values:

**Percentage:** This is the percentage at which the following scalar and color should be applied.

**Scalar:** A scalar (multiplier) on the size of the pie chart/gauge.

**Color:** A color for the pie chart/gauge.

Example:

```
<SUBDATA ColorMap>
//Percentage Scalar Color
85.0000 1.5000 33023
100.0000 2.0000 255
</SUBDATA>
```

---

## Data View

---

### ScreenLayer

This is a list of screen layer names that are hidden in the current view. Each screen layer name is on a separate line of text.

#### Example:

```
<SUBDATA ScreenLayer>  
//These are hidden screen layers  
"pie layer"  
</SUBDATA>
```

---

**Chapter 15 : PowerWorld Simulator Project Files**

---

PowerWorld Project Files provide the user an easy way consolidate and manage files from any application, including PowerWorld binary and display files.

This chapter covers the following:

- Overview of PowerWorld Simulator project files
- Creating a New Project File
- Opening a Project File

---

## Overview of PowerWorld Simulator Project Files

---

Performing a simulation using PowerWorld Simulator sometimes requires using a number of different files. In addition to the case file that stores the model of the system, there may be one or more oneline diagrams depicting various regions of the system. These oneline diagrams might feature [document links](#) that connect to files that were created using other applications, as well as [oneline links](#), which open other oneline diagrams when they are clicked. For these links to function properly, the documents and oneline displays to which they connect must be available. A simulation might also employ a template file that is used to load in a predefined set of solution, environment, and display options. Furthermore, the simulation might make use of data stored in [auxiliary data files](#) to supplement the data stored in the case file. Finally, a simulation might utilize a [script file](#) to perform some automated sequence of tasks, or even to display a movie of system conditions. Having to deal with so many files may make it difficult to transfer the case to another computer, or to share the model with a colleague. PowerWorld Simulator Project files provide a solution.

PowerWorld Simulator Projects have the extension \*.pwp. A project serves as a container for all the files that might comprise a simulation, including the case file, one or more oneline diagrams, a script file, one or more auxiliary data files, and a case template, as well as any other support files you may wish to include. A project is actually a compressed file archive that is compatible with the widely available PKZip and WinZip file compression utilities. It is strongly recommended, however, that you work with project files strictly within the PowerWorld Simulator environment, as Simulator can automatically perform the file compression and extraction functions and process the contents of the included files in a single step. (Note: Simulator performs file compression and extraction using software components available from <http://www.cdrom.com/pub/infozip/> and <http://www.geocities.com/SiliconValley/Orchard/8607/main.html>).

---

**PowerWorld Project Initialization Script**

---

Included in every project file is an initialization script file called OPENPWB.SCP. This is a special type of PowerWorld script file that is used to unload the contents of the project that are used in the simulation. This file must begin with the keyword "INITIALIZATION" and terminate with the keyword "END". Between these two lines, OPENPWB.SCP identifies the name of the case, any oneline diagrams that should be opened immediately, a template file (if any), a script file (if any), and one or more auxiliary data files (if any) to open and read into memory after extracting the files. The file might also contain the keyword "AUTOSTART", indicating that the simulation of the system described by the project file should commence immediately after it is read into memory. The OPENPWB.SCP is generated for you automatically when you save a new project file. Here is an example that loads the case B7FLAT.PWB (in PowerWorld binary format) along with oneline diagram B7FLAT.PWD, the template B7FLAT.PWT, the script B7FLAT.SCP, and the auxiliary file B7FLAT.AUX, and then starts the simulation immediately after all contents are extracted and read:

```
INITIALIZATION
CASE PWB B7FLAT.PWB
ONELINE B7FLAT.PWD
TEMPLATE B7FLAT.PWT
SCRIPT B7FLAT.SCP
DATAFILE B7FLAT.AUX
AUTOSTART
END
```

Again, all projects must contain the file OPENPWB.SCP. If a project does not contain this file, an error message will be shown. Simulator automatically includes an OPENPWB.SCP file with every project it creates. Therefore, unless you try to create a project file outside of Simulator, you will not have to worry about this requirement.

---

## Associating Project Files with Simulator

---

PowerWorld Simulator can automatically open and load the contents of a project if the proper file associations have been made. When Simulator is installed on a computer, it registers the file extension \*.pwp in the Windows registry. Then, whenever you double-click on an icon for a project file or download a project file from a web site, Simulator will start, extract the contents of the project file, and read them into memory. The power system will then be displayed exactly as the author of the project file intended it to be displayed.

If the file association is somehow destroyed (i.e. Windows no longer recognizes that \*.pwp files should be associated with and opened by Simulator), you can re-establish the association manually using the following procedure:

- Windows 95/98/NT
  - o Open Windows Explorer.
  - o Select View > Folder Options from Windows Explorer's main menu.
  - o Switch to the tab labeled "File Types."
  - o Click the button labeled "New Type."
  - o For "Description of Type," specify "PowerWorld Project."
  - o For "Associated Extension," specify "PWP".
  - o For "ContentType (MIME)," specify "application/powerworld\_project."
  - o For "Default Extension," specify ".PWP".
  - o Under "Actions," click the button labeled "New."
  - o Specify the "Action" as "open."
  - o Specify the "Application used to perform action" as the path to the Simulator executable file, followed by a space and the string "%1" (including the quotation marks). For example, if the path of the PowerWorld Simulator executable is "c:\program files\PowerWorld\pwrworld.exe," then you should specify the following string:  
     c:\program files\PowerWorld\pwrworld.exe "%1"
  - o Click OK to close the Action dialog.
  - o Click OK to complete the definition of the file association.
- Windows XP
  - o Open Windows Explorer
  - o Select Tools > Folder Options from Windows Explorer's main menu.
  - o Switch to the tab labeled "File Types"
  - o Click the button labeled New
  - o For "File Extension", enter "PWP", then click OK.
  - o Find the PWP entry in the Registered file types list, then click on it.
  - o Next, with the PWP entry selected, click on Advanced.
  - o In the box to the left of the "Change Icon..." button, type in "PowerWorld Project"
  - o Click on the "New..." button on the right-hand side of this dialog, which will pop up a "New Action" dialog
  - o In the New Action dialog, click "Browse..."
  - o Navigate to and select the PowerWorld program binary (pwrworld.exe); the default location is c:\Program Files\PowerWorld\pwrworld.exe
  - o In the New Action dialog, enter "Open the project" in the field labeled "Action"
  - o Click OK in the New Action dialog and in the Edit File Type dialog
  - o Click Close when you are back to the main File Types dialog.

The \*.pwp file type will now be recognized on your system as being associated with PowerWorld Simulator. Again, you will need to perform this procedure only if the association somehow gets removed.

---

## Creating a New Project File

---

Creating a new project file in Simulator is fairly simple. First, make sure that the case for which you want to create the project is open in Simulator, as well as any oneline diagrams that you wish to open automatically whenever the user opens the project. Then select **Save As Project** from the [Application Button](#). This opens the [Create Project Dialog](#), which you can use to specify the contents of the project file. When you click OK on the Create Project Dialog, the files you identified will be compressed into a single file, along with the [project initialization script](#) that tells Simulator how it should import the contents of the project.

---

## Create Project Dialog

---

The Create Project Dialog enables you to create a new [PowerWorld Simulator Project](#) by specifying the files that should be included in the project. To open the Create Project Dialog, select **Save As Project** from the [Application Button](#). When it opens, the dialog will automatically include the case file, all open oneline diagrams, all documents and onelines linked to the display by [document links](#) and [oneline links](#), the current case template file, and the current script in its list of files to add to the project. You can then use this dialog to add files to or delete files from the list of project contents.

The Create Project Dialog features the following controls:

### Case

Identifies the name of the case file that is currently open in Simulator. The case file may be in PowerWorld binary format, in PTI Version 23 - 29 .raw format, or in GE PSLF .epc format. You cannot change this field, because it must be set to whatever case is currently open.

### Open these oneline diagrams automatically

Lists the names of the oneline diagrams that will automatically open whenever a user opens the project file. This is also a read-only control and lists the names of all oneline diagrams that are currently open in Simulator. Therefore, before you start to create a project using the Create Project dialog, make sure that only those oneline diagrams you wish to open automatically with the project are currently open in Simulator.

### Use this script file

Identifies a [script file](#) that should be opened automatically when the project is open. Scripts are text files that contain commands in [PowerWorld's scripting language](#) that Simulator interprets to perform a predefined set of tasks during the simulation. If a script file is in use when you try to create the project, its name will automatically appear in this text box. To specify a different script file, either type its full path in the text box, or press the adjacent *Browse* button to search for it on your machine.

### Load data from auxiliary files

Lists the [auxiliary data files](#) that should be automatically read when the project is open. Auxiliary data files are text files that contain data that supplement or supersede the information in the case file. You may include as many automatically loading auxiliary data files in the project as you like. To add an auxiliary file to the project contents, click the adjacent Add button and locate the file on your machine. To remove an auxiliary file from the list, select it and click the adjacent Remove button.

### Also include these files

Lists any other files you may wish to include in the project. Use this control to include supplementary documents, pictures, or even other PowerWorld project files, in the project. To add a file to the contents, click Add a File. To remove a file from the list, select it and press the adjacent Remove button.

### Automatically start the simulation after opening the project

Check this button to cause Simulator to start the simulation immediately after opening the project. This saves the user who opens the project the step of switching to run mode and clicking the play button in the [Power Flow Tools](#) ribbon group on the [Tools](#) ribbon tab. This option is particularly effective, for example, if you want to display a "movie" of system conditions when the project is open. By including a script file in the project that tells



Simulator how to modify the system and view over time, you create a project that, when opened or downloaded, animates the scenario you are trying to model.

**Save this project as**

Identifies the name of the project that should be saved. By default, the project name is the same as that of the case except that it has the extension "pwp". To specify a different name, either type it directly in the text box, or click the adjacent Browse button and specify the full path.

When you have finished specifying the contents of the project file, click the **OK, save the project** button. If you wish to cancel the operation without creating the project, click **Cancel**.

---

## Opening an Existing Project

---

To open an existing [PowerWorld Simulator Project file](#), select **Open Project** from the [Application Button](#), and select the name of the project you wish to open. Alternatively, you may select **Open Case** from the [Application Button](#), change the *Files of Type* setting to "PowerWorld Project (\*.pwp)," and select the project you wish to open. Simulator will extract the contents of the project file into the project file's folder and input the information it needs to display the case, oneline diagrams, and other associated files.

---

**Chapter 16 : Tutorials**

---

This chapter contains tutorials on building a new case in Simulator and starting from an existing case in Simulator. They were originally created for use from the on-line help, although they can be used in text format as reproduced here.

- Creating a New Case
- Starting with an Existing Case
- OPF
- Contingency Analysis


---

**Tutorial: Creating a New Case** Page 1 of 13


---

This procedure describes how to create a simple power system model using PowerWorld Simulator. This procedure was developed for use with version 13 of the package. If you have an earlier version, please contact PowerWorld Corporation at [info@powerworld.com](mailto:info@powerworld.com) for information on upgrading, or visit the website at <http://www.powerworld.com>.



To begin, double-click on the PowerWorld Simulator icon.  This starts Simulator. Simulator is used to create new cases, modify existing cases, and (of course) simulate power systems. In this example, we will build a new case from scratch. To create a case from an existing power flow case instead, please see [Creating a Case from an Existing Power Flow File](#).



To create a new case, select **New Case** from PowerWorld icon  in the upper left corner of the program. The screen background will turn white, the default background color for new PowerWorld oneline diagram. Oneline diagrams are used in power system analysis to represent the actual three-phase power system using a single line to represent each three-phase device.

---

**Tutorial: Inserting a Bus** Page 2 of 13
 

---

The most important component of the power system model is the *bus*. Buses are used to represent junction points in the power system where a number of devices are connected together. In building a power system model using Simulator, you will draw buses onto the online diagram, attach devices such as generators and loads to the buses, and connect different buses together with transmission lines and transformers.

To insert a bus:

- Select **Network > Bus** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. This prepares Simulator to insert a new bus.
- Left-click on the online background at the location where you want to place the new bus. This invokes the [Bus Option Dialog](#) (pictured below), which is used to specify the name, orientation, shape, size, width, area, zone, and nominal voltage of the bus, as well as the load and shunt compensation connected to the bus .

- The *Bus Number* field automatically displays '1'. Simulator requires that each bus have a unique number. For convenience, accept the default value. Historically, the *Bus Name* field was limited to eight characters. That limitation is no longer imposed in Simulator. Many users still choose to limit the bus name to eight characters by convention and for ease in converting cases to other formats (such as .epc or .raw). For this example, enter '**One**'.
- Next, check the *System Slack Bus* field, which is located in the Bus Voltage portion of the Bus Information Tab. The Slack Bus is a modeling construct that ensures that the power system has enough generation to meet the load. In other words, the slack bus "picks up the slack" caused by system losses or unbalanced generation and load.
- Click **OK** on the Bus Option Dialog to finish creating the bus and to close the dialog. After the dialog box closes, the new bus appears on the oneline at the location you specified.

---

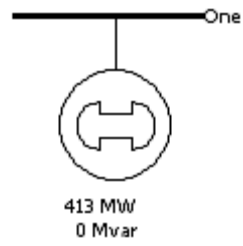
**Tutorial: Inserting a Generator Page 3 of 13**


---

Next we are going to attach a generator to the bus. Generators may be inserted in a manner similar to inserting a bus:

- Select **Network > Generator** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Left-click the bus on the oneline diagram to which you want to attach the generator (for this example, click on the slack bus – bus *One*.) The [Generator Option Dialog](#) (pictured below) will automatically open. The dialog is used to specify the new generator's unit identifier, display size, orientation, MW output and limits, reactive power limits, set point voltage, and cost model.

- Every generator must have a *MW Output* specified when the generator is inserted. Make sure the *MW and Voltage Control* tab is selected. Enter '413' in the *MW Output* Field. Note: the *MW Output* specified for a generator connected to the system slack bus is arbitrary because the generator's true output depends on system load and losses.
- Select the *Display Information* tab. The *Orientation* field is used to specify the direction the generator will extend from the bus. The *Anchored* checkbox forces the generator to move with its specified bus when repositioning the bus on the oneline.
- Click **OK** on the Generator Option Dialog to accept the default values for all other fields. After the dialog box closes, the new generator will appear on the oneline attached to the previously selected bus. The oneline diagram should resemble the image shown below.






---

**Tutorial: Saving the Case** Page 4 of 13

---

To save the work that we have done so far, select **Save Case** from the [Application Button](#), or click on the *Save Case*  button. Before the case is saved, Simulator validates the case to make sure that it does not contain any errors. Results from this validation are displayed in the Message Log display, usually shown in the lower right-hand corner of the display. If the log is not visible, click the **Log** on the Log group under **Tools**. Since we have not yet named the case, the **Save As** dialog is displayed. Enter a file name and select **OK**. By default the case is saved using the [PowerWorld Binary format](#) (\*.pwb). When saving the case in the future, you will not have to reenter its name. Simulator also asks you to supply a name for saving the oneline diagram we have been drawing. The oneline diagram files have a default extension of \*.pwd, which identifies them as PowerWorld Display files. Supply the same name as you gave to the case. Note that, because the case and the oneline are stored in separate files, multiple onelines can be assigned to the same case, and the same oneline can be used by many cases.

---

**Tutorial: Entering a Second Bus with Load** **Page 5 of 13**


---

To enter the second bus:

- Select **Network > Bus** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Click on the oneline diagram somewhere to the right of the first bus. In the Bus Options Dialog (pictured below) leave the bus number at the default value of 2, and enter the name 'Two' in the *Bus Name* field.
- We will model a 200 MW, 100 Mvar load at the bus. Select the *Attached Devices* tab. Under the *Load Summary Information* heading enter '200' in the *Base MW* field and '100' in the *Base Mvar* field.
- Click **OK** to accept all other default values, close the Bus Options Dialog, and insert the bus.

**Bus Options**

Bus Number: 2 (Find By Number, Find ...)

Bus Name: Two (Find By Name)

Nominal Voltage: 138.0 kV

Labels ...: no labels

	Number	Name
Area	1	1
Zone	1	1
Owner	1	1
Substation		

Bus Information | **Display** | Attached Devices | Custom | Fault Parameters

**Load Summary Information**

Base MW: 200.0  
Base Mvar: 100.0  
Add or Edit Bus Load

**Shunt Admittance**

G (MW): 0.0  
B (Mvar): 0.0

OK Save Cancel

At this point, the oneline diagram does not show the load at bus 2, even though it is represented in the power system model (you can confirm this by right-clicking on bus 2, selecting *Bus Information Dialog* from the resulting local menu, and inspecting the *Load Summary Information* fields again).

To draw the load on the oneline diagram:

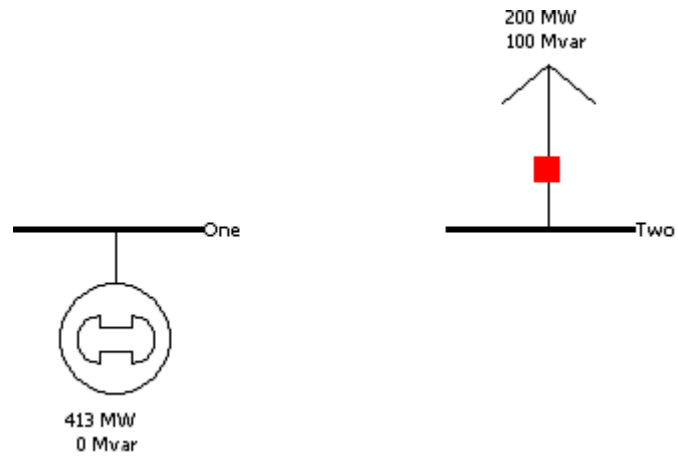
- Select **Network > Load** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Left-click in the center of this bus. The Load Options Dialog box (pictured below) automatically opens. The *Constant Power MW* and *Mvar* fields confirm that the load is 200 MW and 100 Mvar. In addition to constant power loads, Simulator also allows the modeling of voltage dependent loads.

- Select **Up** in the *Orientation* field under the *Load Information* tab to make the load point up. Verify that the anchored box is checked to force the load to move with the selected bus.
- Click **OK** to accept the default values for all remaining fields, close the Load Options dialog, and insert the load. A circuit breaker symbol is automatically included with each load.

To move objects on the oneline:

- Left-click on the desired object. Drag and drop the object to the new location by holding the left mouse button down while moving the mouse. Note: you can also move all objects on the oneline simultaneously by left-clicking on the diagram (not on a specific object) then dragging and dropping in the desired location.
- To move bus 2, left click on bus 2 (not on the attached load). Drag the bus to a new location. Note that the load moves with the bus because it is [anchored](#). You can change the location of attached devices connected to a bus, such as generators and loads, by the same procedure.

The oneline diagram should now resemble the image shown below.



---

**Tutorial: Inserting a Transmission Line Page 6 of 13**


---

Transmission lines are used to connect buses together. To insert a transmission line:

- Select **Network > Transmission Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Left-click at the point where you want the new line to originate. This point is usually located on one of the proposed line's terminal buses. For this example, originate the line at bus One.
- Transmission lines and transformers are drawn as a series of line segments. Without holding down the mouse button, drag the mouse up. Notice that a line segment connected to the point of origin will follow your mouse movements. To terminate a line segment, click the left mouse button. Each time you click the mouse to terminate a line segment, a new vertex is defined for the line. To draw the next line segment, move the mouse to the desired location of the next vertex. Note: the vertices may later be moved or deleted to reshape the line. To create curved lines, hold the left mouse button down while dragging.
- To terminate the final line segment and conclude drawing the line, double click the left mouse button at the desired termination point (bus Two for this example). The termination point is usually the transmission line's other terminal bus.
- The [Transmission Line/Transformer Dialog](#) automatically appears (shown below). The dialog should already contain a 1 in the *From Bus Number* field and a 2 in the *To Bus Number* Field. If not, you probably did not have the cursor directly on the bus when you were drawing the line. If this is the case, simply enter the correct bus numbers in the corresponding fields.

- The *Series Resistance*, *Series Reactance*, and *Shunt Charging* fields are used to enter the per unit parameters associated with the line. The *Shunt Charging* field contains the total per unit charging capacitance for the line. Enter **0.02** in the *Resistance* field, **0.08** in the *Reactance* field and **0.1** in the *Shunt Charging* field.

- The *Limit (MVA)* fields contain the MVA ratings for the line; enter a value of 1000 in the *Limit A (MVA)* field.
- Click **OK** to accept all remaining field default values, close the Transmission Line/Transformer Dialog, and insert the new line.

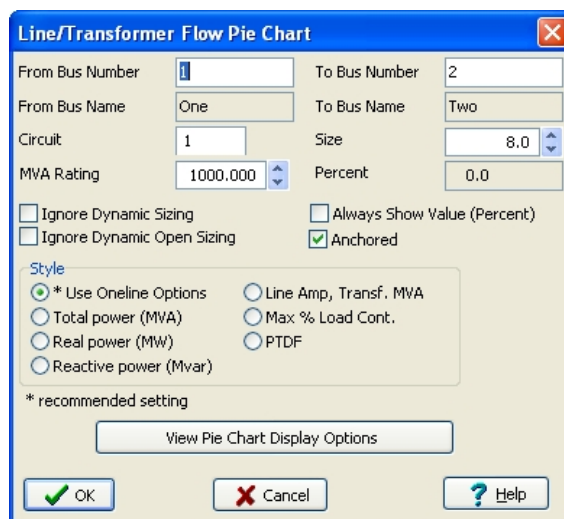
By default, the transmission line is anchored to both terminal buses. If you try to move bus 2, the transmission line should move with it.

---

**Tutorial: Inserting a Line Flow Pie Chart on a Transmission Line** Page 7 of 13
 

---

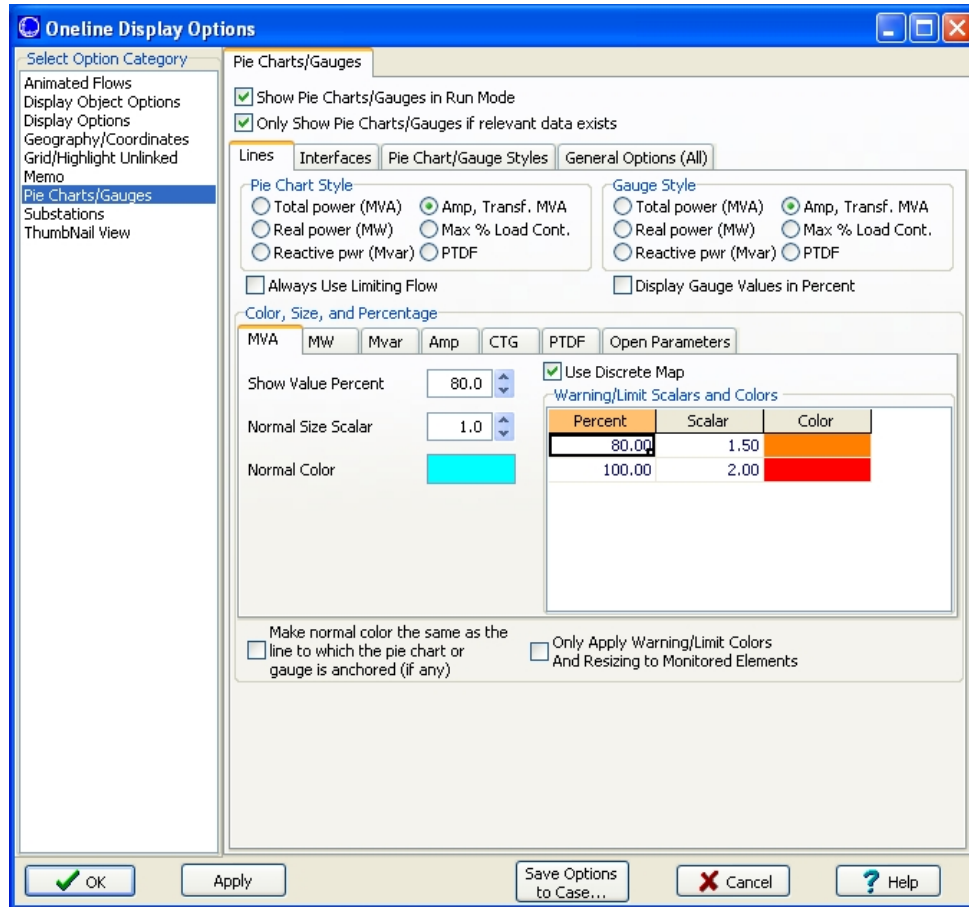
When the line is drawn it automatically it has a line flow pie chart included. You can include additional line flow pie charts by selecting **Pies/Gauges > Line Flow Pie Chart** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab, and then clicking near a line. The Line/Transformer Flow Pie Chart dialog box appears (shown below). Make sure that the From Bus and To Bus have the correct numbers, that the MVA rating is correct, and that Anchored is checked. You may change the size of the pie chart by typing in a value or using the arrows.



The dialog box is titled "Line/Transformer Flow Pie Chart". It contains the following fields and options:

- From Bus Number:** 1
- To Bus Number:** 2
- From Bus Name:** One
- To Bus Name:** Two
- Circuit:** 1
- Size:** 8.0
- MVA Rating:** 1000.000
- Percent:** 0.0
- ☐ Ignore Dynamic Sizing
- ☐ Always Show Value (Percent)
- ☐ Ignore Dynamic Open Sizing
- ☒ Anchored
- Style:**
  - ☒ \* Use Online Options
  - ☐ Line Amp, Transf. MVA
  - ☐ Total power (MVA)
  - ☐ Max % Load Cont.
  - ☐ Real power (MW)
  - ☐ PTDF
  - ☐ Reactive power (Mvar)
- \* recommended setting**
- View Pie Chart Display Options** (button)
- OK** (button)
- Cancel** (button)
- Help** (button)

Color and behavior of pie charts are set in the *Online Display Options* display; right-click anywhere in the background of the online, select *Online Display Options* from the pop-up menu, then select the *Pie Charts* tab (pictured below).





---

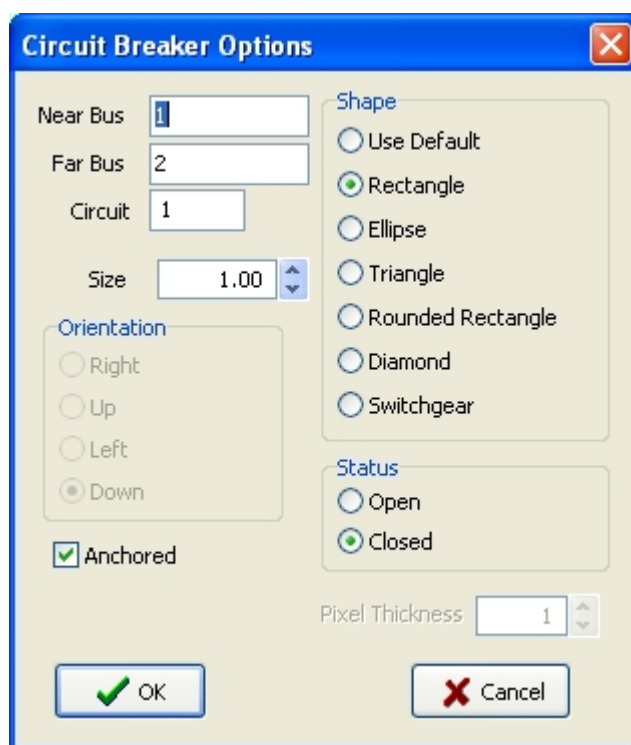
**Tutorial: Inserting Circuit Breakers Page 8 of 13**

---

Circuit breakers are used to control the status of the line. (If the line already has circuit breakers at each end, then Simulator has been instructed to insert circuit breakers automatically. You can configure this option from the [Default Drawing Options Dialog](#)).

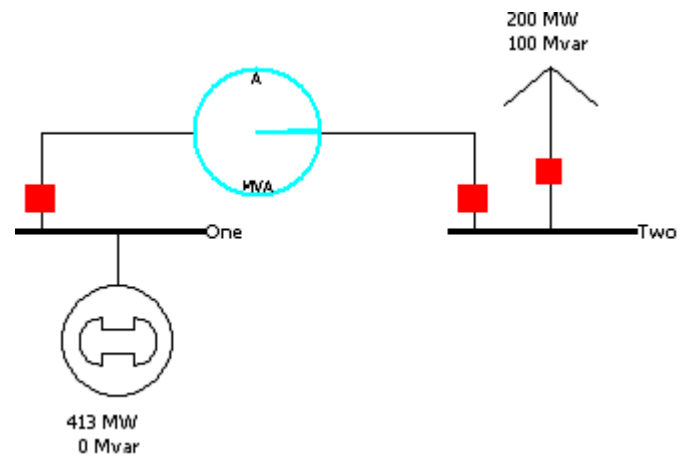
To insert a Circuit Breaker:

- Click somewhere on the line near bus One then select **Indication > Circuit Breaker** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab, then click on the line near bus One. You should immediately see the Circuit Breaker Options dialog (shown below) with the From Bus and To Bus fields correctly set to '1' and '2'. If they are '0', enter the correct value. Set the Size field to '1' (you can either enter a 1, or use the spin arrows in change the value).
- Click **OK** to insert the circuit breaker.



In Simulator, the location of the circuit breaker does not matter, because changing the status of the circuit breaker changes the status of the entire line. However, since most transmission lines have circuit breakers at each end, we will also place a circuit breaker near bus 2. To accomplish this, repeat the above process near bus 2.

**Save your case.** Your oneline should now look similar to the image below.



---

**Tutorial: Inserting a Transformer Page 9 of 13**


---

To insert a transformer we first need to insert a bus at a different voltage level. Insert a new bus in the bottom of the oneline, named **Three**, and enter **69 KV** for its nominal voltage in the Bus Options dialog box.

To insert a transformer between buses 2 and 3:

- Either go to **Network > Transformer** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Click on bus 2, and then draw a line to bus 3 as you did for a transmission line. The [Transmission Line/Transformer Dialog](#) automatically appears.

**Transmission Line/Transformer Options**

From Bus: 2 To Bus: 3 Circuit: 1

Number: Two Name: Three

Area Name: 1 (1) 1 (1)

Nominal kV: 138.0 69.0

Labels: no labels

Find By Numbers Find By Names Find ...

☒ From End Metered ☒ Default Owner (Same as From Bus)

Display Parameters Transformer Control Fault Info Owner, Area, Zone, Sub Custom

Status: ☐ Open ☒ Closed

Length (mi): 0.00

Calculate Impedances >

Per Unit Impedance Parameters:

Series Resistance (R)	0.02000
Series Reactance (X)	0.08000
Shunt Charging (B)	0.1000
Shunt Conductance (G)	0.0000
Magnetizing Conductance	0.000000
Magnetizing Susceptance	0.000000
<input type="checkbox"/> Has Line Shunts	Line Shunts

MVA Limits:

Limit A	1000.000
Limit B	0.000
Limit C	0.000
Limit D	0.000
Limit E	0.000
Limit F	0.000
Limit G	0.000
Limit H	0.000

Convert Transformer to Line

OK Save Cancel Help

- On the **Parameters** tab, enter **0.02** for *Series Resistance*, **0.08** for *Series Reactance*, **0.1** for *Shunt Charging* and **1000** for *Limit A (MVA)*.
  - Select the **Transformer Control** tab. Note that the Off-nominal Turns Ratio displays **1.000**. The true transformer turns ratio does not need to be specified as it is automatically determined by the ratio of nominal voltages between the *From Bus* and *To Bus*. The *Off-nominal Turns Ratio* is used to adjust the transformer tap setting on per-unit values of bus voltages as referenced to their respective base values for per-unit calculation.
  - Click **OK** to accept the default values, close the dialog and insert the transformer.
  - Repeat this procedure to add a transformer between buses 1 and 3.
- Add a 400 MW, 200 Mvar load to bus 3:
- Right-click on bus 3 and select **Bus Information Dialog**. Select the **Attached Devices** tab. Click **Add or Edit Bus Load**.
  - Under *Constant Power* enter **400** in the *MW Value* field. Enter **200** in *Mvar Value* field. Click **OK**.
  - Note that *Base MW* and *Base Mvar* display the respective values. Click **OK**.

The load is now attached to the bus even though it is not displayed on the oneline. To display the load as an object on the oneline, you can either use the **Auto Insert** feature or follow the procedure utilized earlier. To **Auto Insert** the load:

- Select **Auto Insert > Loads...** from the [Quick Insert](#) ribbon group on the [Draw](#) ribbon tab. The *Automatic Insertion of Loads* dialog opens automatically.
- Click **OK** to accept the default values and insert the load object on the oneline.

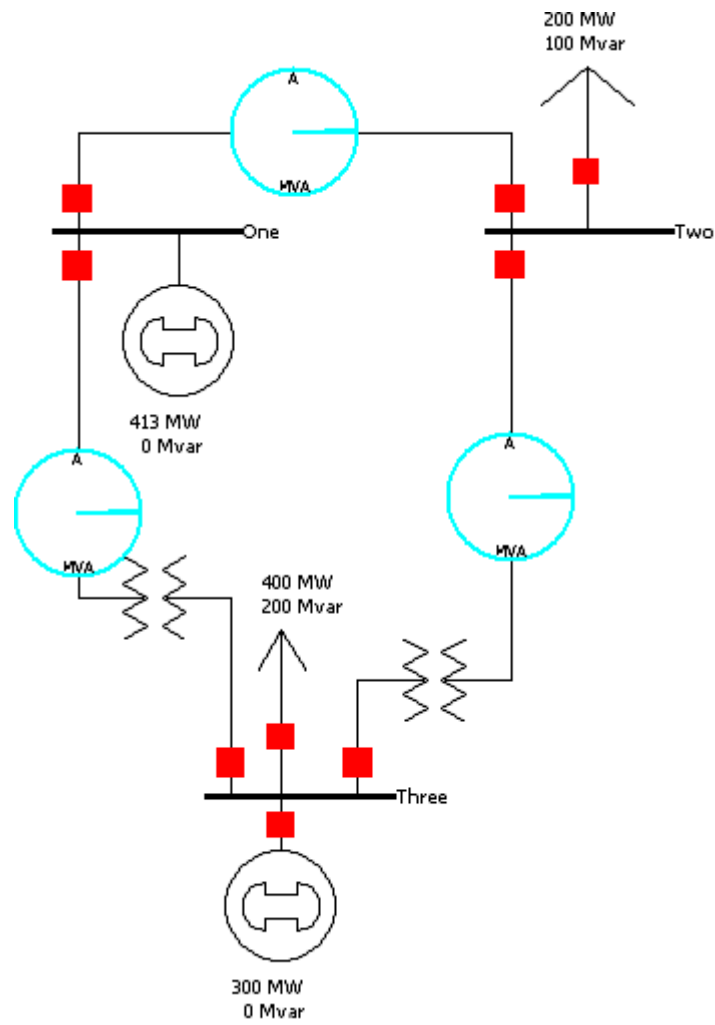
Note that Lines, Loads, Interfaces, Generators and Switched Shunts can all be inserted as objects on a oneline using the **Auto Insert** tool if a record already exists for the device.

- Left-click and drag the load to the desired location on bus 3. Note: you can resize the bus object on the oneline by left-clicking on the bus then dragging either end-point vertex to the desired bus size.
- Right-click on the load and select **Load Information Dialog**. You can change the load orientation and verify all load parameters in this dialog.
- Click **OK**.

Next we are going to attach a 300 MW generator to the bus 3:

- Select **Network > Generator** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Left-click on bus 3. The [Generator Option Dialog](#) opens.
- Select the *Power and Voltage Control* tab. Enter '300' in the *MW Output* Field.
- Click **OK** on the Generator Option Dialog to accept the default values for all other fields. After the dialog box closes, the new generator appears on the oneline attached to bus 3.

Your oneline should now resemble the image shown below.



---

**Tutorial: Inserting a Switched Shunt Page 10 of 13**


---

Switched shunts usually consist of either capacitors to supply reactive power (in MVAR) to the system, or reactors to absorb reactive power. The switched shunts are represented by a number of blocks of admittance that can be switched in a number of discrete steps. If at least one block is in service, the shunt is said to be online. The shunt's corresponding circuit breaker is used to determine and / or toggle the switched shunt's status.

- To insert a switched shunt at bus 3, select **Network > Switched Shunt** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Click on or near bus 3. The *Switched Shunt Options* dialog box appears (pictured below).
- Verify that the bus number is 3; if it is not, change it.
- Enter **10** for the *Nominal Mvar*.
- Click **OK** to accept the default values of the remaining fields, close the dialog, and insert the switched shunt.

**Switched Shunt Options**

Bus Number: 3 Find By Number

Bus Name: Three Find By Name

Shunt ID: 1 Find ...

Labels ...

Area: Change 1 1

Zone: Change 1 1

Substation:

Display Parameters Control Parameters Fault Parameters Custom

Nominal Mvar:

Control Mode:

- ☒ Fixed
- ☐ Discrete
- ☐ Continuous
- ☐ Bus Shunt (Fixed)

Control Regulation Settings:

- ☒ Voltage High Value: 1.0000 Low Value: 0.9900 Target Value: 1.0000
- ☐ Generator Mvar

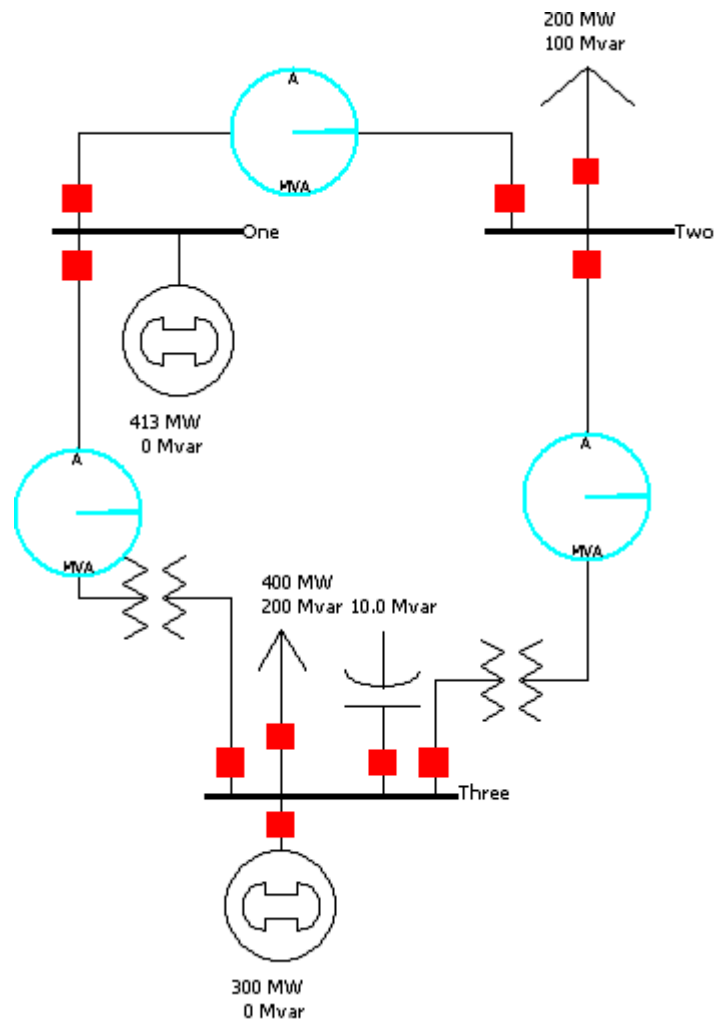
Reg. Bus #: Target Value: 1.0000

Switched Shunts Blocks:

Number of Steps	Mvars per Step
1	

OK Save Cancel Help

Your oneline should now look similar to the image below.





---

**Tutorial: Inserting Text, Bus and Line Fields** Page 11 of 13
 

---

Informational fields can be entered directly on the oneline to allow for ease of monitoring when a case is animating.

To insert a general text field:

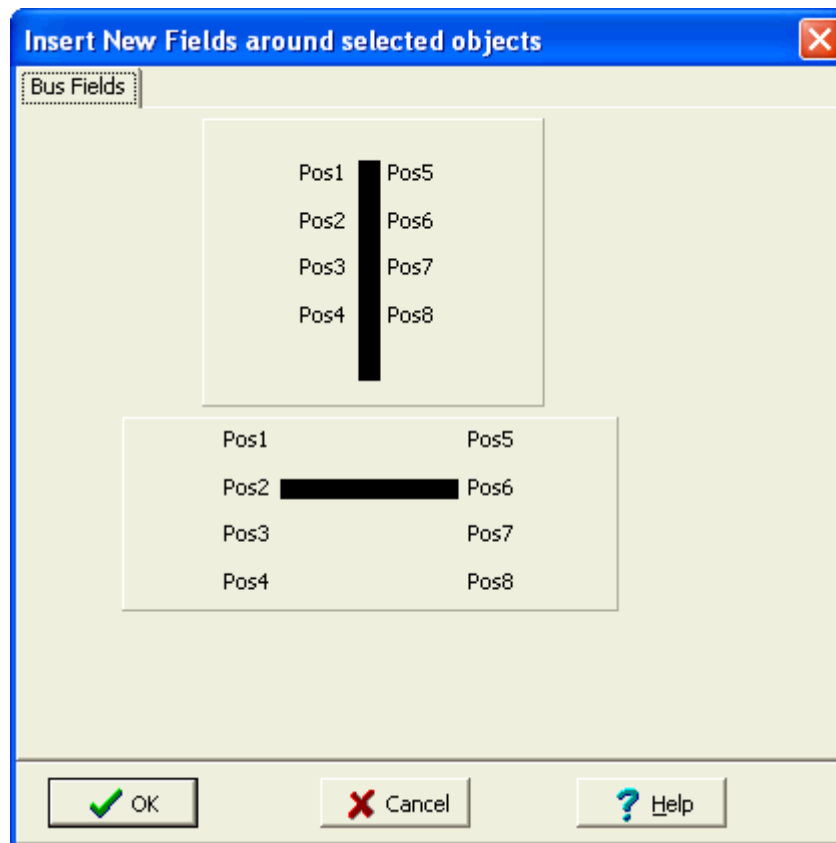
- Select **Background > Text** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Left-click on the oneline in the desired text location to bring up the *Text Object Dialog*. (For this example, left-click in the top center of the oneline.)
- Type the string "First Case" in the *Enter the text* field. Click **OK**.
- To format the text, ensure the text is selected on the one-line then select  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab. This displays the [Font Tab](#) of the [Format Selection Dialog](#).
- Set the font size to **26** and the font color to **blue**.
- To change the text background color, select  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab, which summons the Line/Fill Options Tab . If the format menu is already open, click on the [Line/Fill Options Tab](#) of the Format Selection Dialog. Check the *Use Background Fill* box to give the text a white background, and then click **OK**.
- Inserted text can be moved using the same method as any other object on the oneline

Fields can also display object-related quantities. By default, Simulator has inserted the bus names, generator and load MW and MVAR, and switched shunt MVAR. For this example, we will add a Bus Voltage Magnitude field to each bus and fields showing the power flow on the transmission line and the transformers. Note that object fields can be formatted just like text fields by using the Format menu.

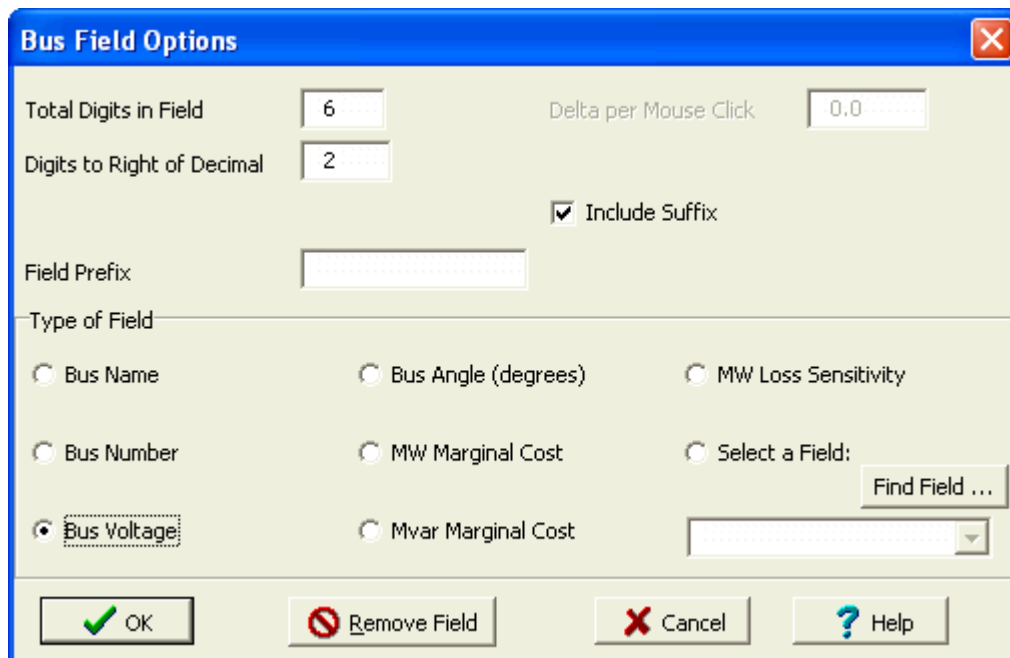
To add additional fields to the display of a particular bus:

- Right-click on the bus to bring up the bus' local menu.
- Select *Add New Fields Around Bus* from the local menu. This opens the [Insert Bus Fields Dialog](#) (pictured below). You may add up to 8 fields per bus. Select the position where you would like to add the new field (position 5 for this example) then click **OK**.





- This opens the Bus Field Options dialog (shown below); select the field **Bus Voltage** to add in the selected position and click **OK**.



- The parameter and position are displayed as highlighted in the Insert New Fields dialog. Click **OK**. Note that the specified bus field has been added to the oneline diagram.
- Repeat this procedure for the other two buses. If necessary, you may move fields manually with the mouse.

Fields can also be inserted using the **Background > Field** option from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.

Next, we will insert fields showing the power flow at each end of the transmission line.

- Select **Field > Transmission Line Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.

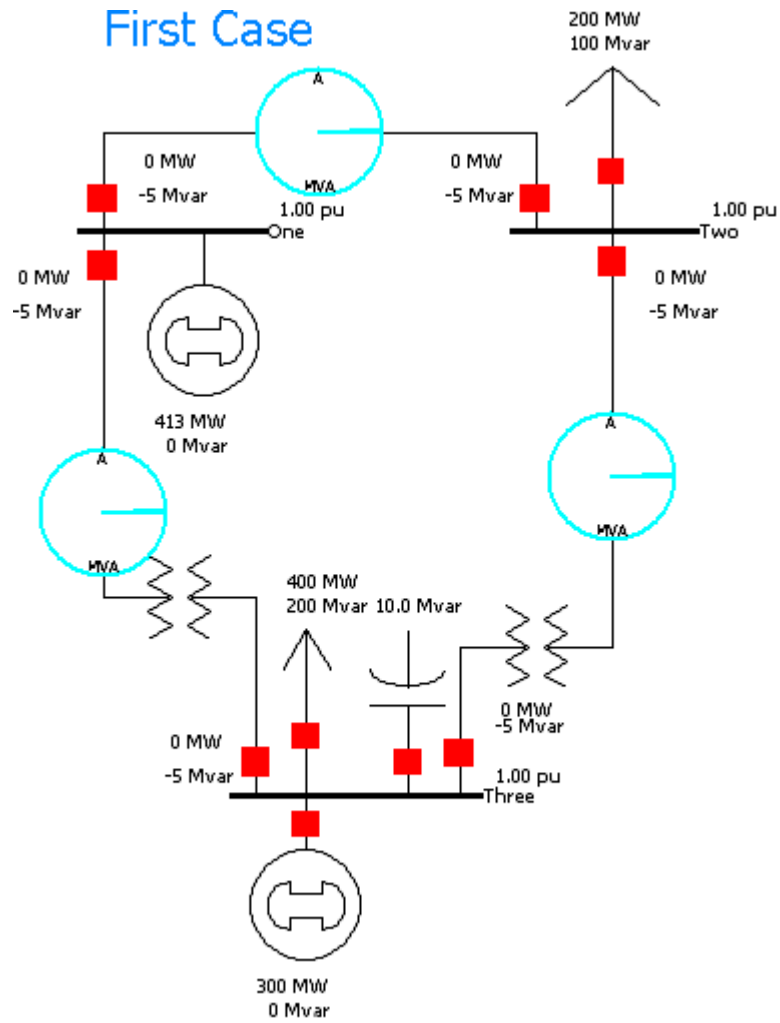
Line fields show information about transmission lines and transformers. For line fields, flow is always specified at an end of the transmission line or transformer. The end is normally determined automatically by the insertion point.

- Left-click near both bus **1** and the transmission line between buses 1 and 2 in the location you want the power flow text to appear. The *Line Field Options* dialog (shown below) opens automatically.
- The *Near Bus* and *Far Bus* fields should show **1** and **2** respectively. If they do not, enter the correct values.
- Select **MW Flow** then click **OK**. The field is displayed on the oneline in the location you specified. Note that the field can be moved and formatted as previously discussed.
- Select **Field > Transmission Line Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab.
- Left-click near both bus **2** and the transmission line between buses 1 and 2 in the location you want the power flow text to appear. The *Line Field Options* dialog (shown below) opens automatically.
- Now the *Near Bus* and *Far Bus* fields should show **2** and **1** respectively. If they do not, enter the correct values.
- Select **MW Flow** then click **OK**.
- Repeat the procedure to insert **Mvar Flow** fields for the two locations.

We also desire to monitor the MW and Mvar flows on the lines joining buses 1 and 3 and buses 2 and 3 via the transformers. The same commands are used as those used to insert fields for the transmission line.

- Repeat the above steps to insert **MW** (and **Mvar**) **Flow** fields on the lines joining buses 1 and 3 and buses 2 and 3 via the transformers.
- At this point, your first oneline diagram should resemble the one shown below.

## First Case




- Save the case.

---

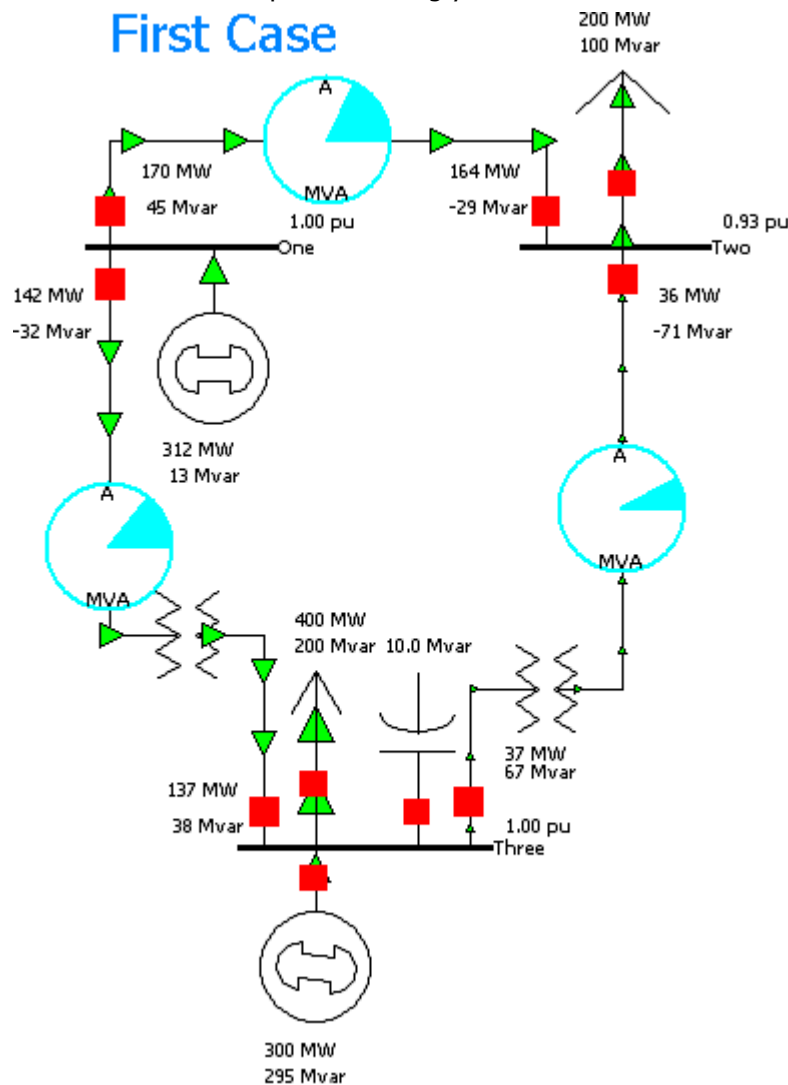
**Tutorial: Solving the Case Page 12 of 13**


---

To solve a case, you must be in run mode:

- Click on **Run Mode** button in the **Mode** ribbon group. Note that if the case has [validation errors](#), a warning will appear. You will need to rectify the problems before you can enter Run Mode.
- Press the **Play**  button in the **Power Flow Tools** ribbon group on the **Tools** ribbon tab to begin the simulation. Alternatively, to perform a single Power Flow Solution, click the **Single Solution - Full Newton** button in the **Power Flow Tools** ribbon group on the **Tools** ribbon tab. Your case should look similar to the case shown below. If it does, congratulations! You have completed building your first case.

### First Case



Try clicking on the load circuit breaker to toggle the load's status. A solid red circuit breaker indicates that it is closed, a hollow green box indicates it is open. While the simulation is running, click on the circuit breakers and note the nearly instantaneous change in system flows. If the Log window is visible, you will get a "backstage" view of what Simulator is doing.

Feel free to close the log. To re-open the log, click the **Log** button in the **Log** ribbon group on the [Tools](#) ribbon tab.

With the load circuit breaker closed, open the circuit breaker between bus 3 and its connected generator. Now open any of the transmission line or transformer circuit breakers.

Congratulations, you've just blacked-out your case!

---

**Tutorial: Adding a New Area** Page 13 of 13
 

---

Next, we will create a second operating area for the case. Large interconnected systems usually have a number of control areas, with each control area responsible for the operation of a particular part of the system. Often, a single control area corresponds to a single owner (such as an investor-owned utility), but it is not unusual for a single control area to have more than one owner. Control areas are connected to neighboring areas through tie lines. A tie line is a transmission line that has one end in one control area and the other end in another. The total amount of power flowing out of a control area is the algebraic sum of the power flowing out on all the area's tie lines. Each control area is responsible for procuring enough power to meet its own load plus losses. The control area can get this power either by generating it itself, or by buying it from another area. This ability to buy and sell power (i.e., power transactions) is one of the principal advantages of interconnected operation.

To create another control area:

- Change to **Edit Mode**. Right click on bus 3 and select **Bus Information Dialog** from the drop down menu.
- Entering a number for an area that does not already exist automatically creates a new area. Enter '2' in the *Area Number* field. Enter 'TWO' in the *Area Name* field. Click **OK**.
- To verify that the case now has two areas, select **Case Summary** from the **Case Data** ribbon group on the [Case Information](#) ribbon tab. The [Case Information Displays](#) allow you to view the entire case using non-graphical displays. The [Case Summary](#) dialog (shown below) shows the number of buses, generators, lines/transformers, and control areas in the case. You cannot modify any of these values.

**CaseSummary for Current Case**

Number of Devices in Case			
Buses	3	Series Capacitors	0
Generators	2	2 Term. DC Lines	0
Loads	2	N-Term. DC Lines	0
Switched Shunts	1	Areas	2
Trans. Lines (AC)	3	Zones	1
LTCs (Control Volt)	0	Islands	1
Phase Shifters	0	Interfaces	0
Mvar Controlling	0	Injection Groups	0

Case pathname: C:\Program Files\PowerWorld\Simulator\Cases\firstcase.PW

Case Totals (for in-service devices only)		
	MW	Mvar
Load	600.0	300.0
Generation	611.7	308.3
Shunts	0.0	-10.0
Losses	11.8	18.3

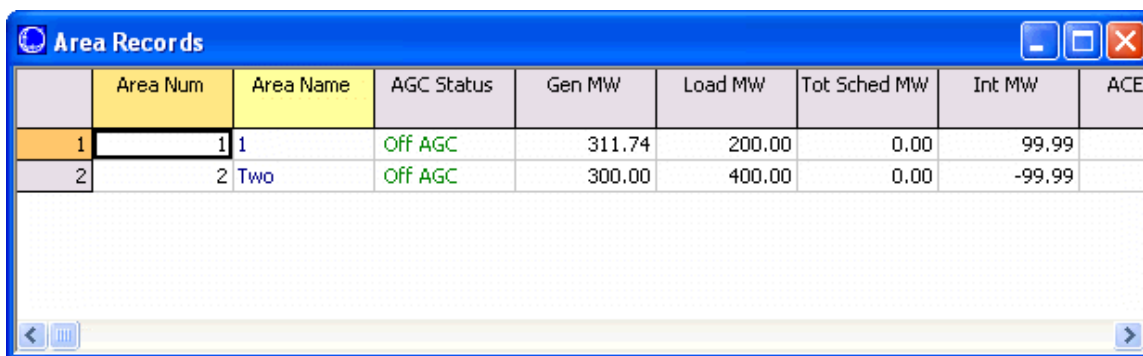
Generator Spinning Reserves	
Positive [MW]	Negative [MW]
1388.3	611.7

Slack Buses:  
One (1); in Area 1 (1)

Buttons: OK, Print, Help

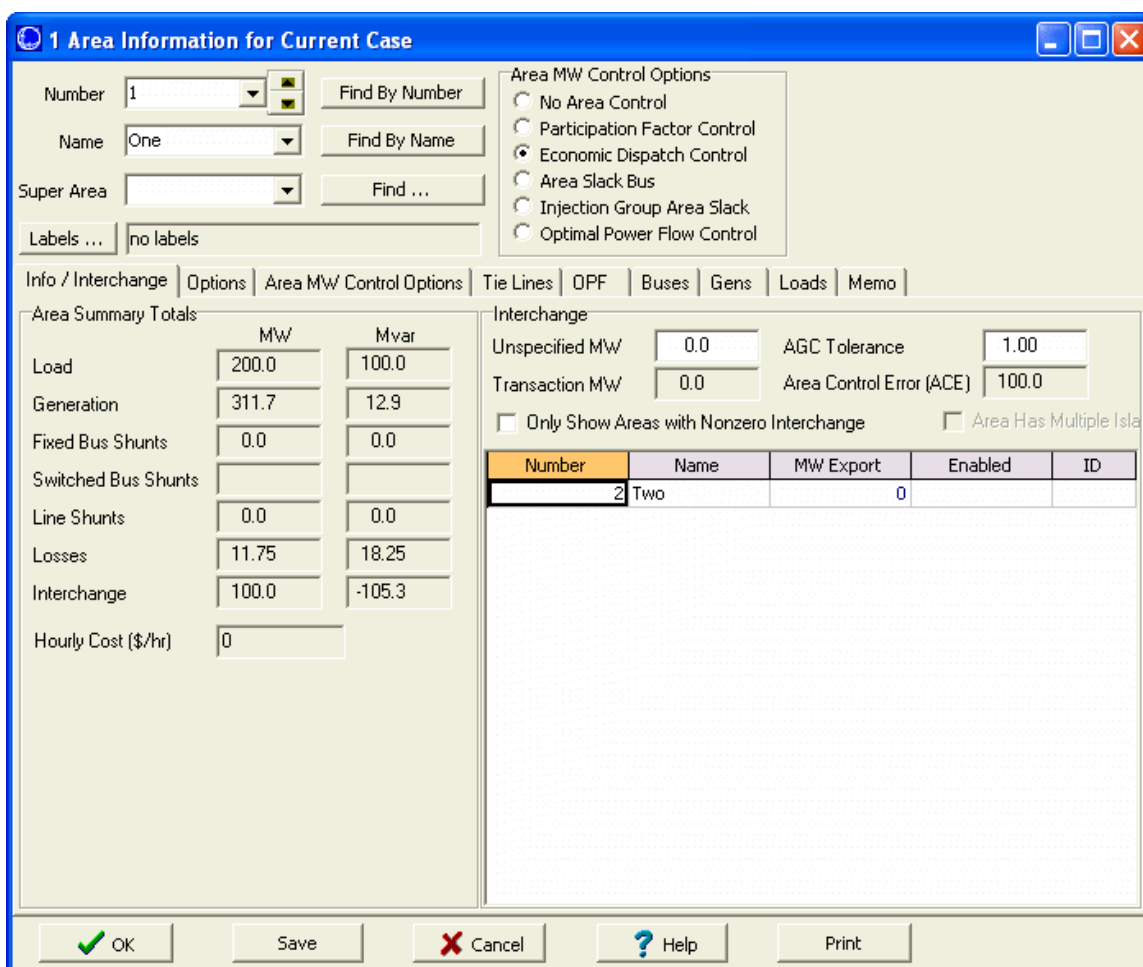
Now we will make sure that both of the control areas are initially set as being on automatic generation control (AGC). AGC insures that the generation in the area is equal to the load plus losses plus and scheduled transactions.

- Select **Case Information > Aggregation > Areas**. The Area display (pictured below) provides a convenient summary of all the control areas in the case. Similar displays exist for buses, generators, lines/transformers, etc.



	Area Num	Area Name	AGC Status	Gen MW	Load MW	Tot Sched MW	Int MW	ACE
1	1	1	Off AGC	311.74	200.00	0.00	99.99	
2	2	Two	Off AGC	300.00	400.00	0.00	-99.99	

- Right-click anywhere on the record for the first area and select the *Show Dialog* option. The [Area Display Dialog](#) is shown (as below).



**1 Area Information for Current Case**

Number: 1 Find By Number

Name: One Find By Name

Super Area: Find ...

Labels ...: no labels

Area MW Control Options:

- ☐ No Area Control
- ☐ Participation Factor Control
- ☒ Economic Dispatch Control
- ☐ Area Slack Bus
- ☐ Injection Group Area Slack
- ☐ Optimal Power Flow Control

Info / Interchange | Options | Area MW Control Options | Tie Lines | OPF | Buses | Gens | Loads | Memo

Area Summary Totals:

	MW	Mvar
Load	200.0	100.0
Generation	311.7	12.9
Fixed Bus Shunts	0.0	0.0
Switched Bus Shunts		
Line Shunts	0.0	0.0
Losses	11.75	18.25
Interchange	100.0	-105.3
Hourly Cost (\$/hr)	0	

Interchange:

Unspecified MW: 0.0 AGC Tolerance: 1.00

Transaction MW: 0.0 Area Control Error (ACE): 100.0

☐ Only Show Areas with Nonzero Interchange ☐ Area Has Multiple Islands

Number	Name	MW Export	Enabled	ID
2	Two	0		

OK Save Cancel Help Print

- Change the Area *Name* to 'One' and set the AGC Status under *Area MW Control Options* to **Economic Dispatch Control**. Select **SAVE** to save this information.
- Next click on the up arrow next to the right of the Area Number field. This displays the Area Record dialog for the next area. Set the AGC Status to **Economic Dispatch Control**, and select **OK** to save your changes and close the dialog.
- Note the *AGC Status* field in the **Area Records Display** now shows **ED** in the *AGC Status* fields. Close the Area Records display.





---

**Starting with an Existing Case** Page 1 of 15

---

This procedure describes how to create a power system model from an existing power flow file using PowerWorld Simulator. This tutorial assumes that you have at least some familiarity with PowerWorld Simulator. If you need a more general introduction, please see [Creating a New Case tutorial](#).

PowerWorld cases can be easily created from existing power flow cases stored in PTI RAW versions 23-30, GE PSLF text format (EPC version 11.X and later), and IEEE common format.

Simulator provides a static model of a power system. The power flow data is a subset of the system model. For some studies this model is sufficient. For other studies the model needs to be augmented by adjusting: generator cost information, the reactive capability curve, PowerWorld Simulator case options, interface definitions, injection group definitions, contingency definitions, the time variation of the load, etc....

To begin, double-click on the PowerWorld Simulator icon. This starts Simulator. In this example we will be building a case from an existing power flow file.

---

**Tutorial: Loading an Existing Power Flow File Page 2 of 15**

---

Once Simulator has been started, set Simulator to Edit Mode by clicking on the **Edit Mode** button. Select **Open Case** from the [Application Button](#). An **Open** dialog box will appear on the screen. To select a power flow file, click on the **Type of File** field in the lower left hand corner of the dialog box. A list of file formats will appear. Choose the appropriate format, and the available files of the type selected will appear in the box above **Type of File**. Choose the desired file from the list of available files and select *OK*. *In this example, we will be building a oneline for the 3990 bus MAIN 1998 summer case saved in PTI version 23 format as mdb98s.raw. When prompted to create a oneline, select 'yes'.* Your display should immediately turn white. This shows the blank background upon which you will be drawing the oneline diagram for your case. Onelines are used in power system analysis to represent the actual three-phase power system using a single line to represent each three phase device. The information from the power flow file is now accessible to Simulator.

---

**Tutorial: Case Information Displays Page 3 of 15**


---

Simulator presents many different [Case Information Displays](#) to provide a text-based view of the case. For example, to view or modify the description of the case, select **Case Description** from the **Case Data** ribbon group on the [Case Information](#) ribbon tab. The **Case Description Dialog** should appear, showing the text description of the power flow case. In PowerWorld, this text description may be arbitrarily long.

You can also obtain a case summary by selecting **Case Summary** from the **Case Data** ribbon group on the [Case Information](#) ribbon tab. The summary dialog will appear. It provides a summary of the case and of the total case load and generation. None of the fields in this dialog can be changed, as they are intended for informational purposes only.

The Area/Zone Filters information display is another important article. The [Area/Zone/Owner Filters](#) feature restricts the contents of other case information displays to certain areas or zones. This is particularly useful for large cases. Open the Area/Zone Filters display by selecting **Area/Zone Filters** from the **Case Information** ribbon group on the [Case Information](#) ribbon tab. As with all Case Information Displays, you can click on a column heading to sort the list on a particular field; click on that same column heading again to reverse the sort order. To change a particular area's area/zone filter status, simply left-click on the area's area/zone filter status field. To change the filter status for all areas in the case, right-click on the display to bring up its local menu (all case information displays have a local menu), and select either **Toggle All Yes** or **Toggle All No**. For our example, set all areas to 'no' except WUMS.

Another important text-based display is the Power Flow List. The [power flow list](#) shows the complete power flow information for all areas whose area/zone filter is set to 'yes'. To display the power flow list, select **Power Flow List** from the **Case Data** ribbon group on the [Case Information](#) ribbon tab. The power flow list has other options that can be accessed by right-clicking anywhere on the list. If you wish to view more details about a particular device, you can do so by holding down the **Ctrl** key and left-clicking on the device. To move through the list of buses, you can use the arrow keys or the scroll bar. If you are viewing a particular bus and you wish to view a bus connected to it, double-click on the line connecting the two buses to view the flows at the other bus. Note that some of the transmission lines have a circuit identifier of '99'. Whenever a '99' is used for an identifier, it usually means that the transmission line is an equivalent line.

A shorter version of the power flow list can often be more useful. Such a display is offered by the Quick Power Flow List. The [Quick Power Flow List](#) is similar to the power flow list except that it shows flows at individual buses or a set of buses, regardless of area/zone filter settings. You can view flows at any bus in the case. To display the quick power flow list, select **Quick Power Flow List** from the **Case Data** ribbon group on the [Case Information](#) ribbon tab. Identify the buses of interest in the **Bus Number** field by entering the number of either a single bus, a set of buses separated by commas, or a range of buses specified using a hyphen. Again you can double-click to move to a desired bus, or choose **Show Object Dialog** from the display's local menu to see the information dialog for any object.

---

**Tutorial: Solving the Case Page 4 of 15**

---

To solve the power flow case we have been using in this example, press the **Single Solution - Full Newton** button in the [Power Flow Tools](#) ribbon group on the [Tools](#) ribbon tab, or press the Single Solution icon on the [Quick Access](#) toolbar. You may wish to show the Message Log before you solve the case so as to monitor the solution process. When you choose to perform a Single Solution, the application automatically switches to Run Mode if it is not already there. The system has initial mismatches because of voltage truncation in the power flow file. The case should converge quickly, perhaps in 2 or 3 iterations. After the single solution has been performed, Simulator now has the solved power flow in memory, and you are ready to build the oneline.

It is not necessary that the power flow case be solved before you create the oneline. However, we recommend that you solve the existing case first to make sure that it is valid.

---

**Tutorial: Building Onelines Page 5 of 15**

---

PowerWorld Simulator makes the power system case easier to analyze by presenting results visually using a oneline diagram. You do not need to represent every bus in the power flow model on the oneline; a oneline diagram need be created only for the desired portion of a system under study. Simulator can automatically link the constructed oneline diagram to the existing power system model. The following sections of this tutorial will discuss placing various visual components onto a oneline diagram. In all of the following sections the actions described will relate to the aforementioned MAIN example.

---

**Tutorial: Entering a Bus Page 6 of 15**

---

The most important component of the power system model is the *bus*. Buses are used to represent junction points in the power system where a number of devices connect together. To build a oneline diagram, you draw the buses, attach devices such as generators and loads to the buses, and connect the buses together with transmission lines and transformers.

To begin entering devices onto the blank oneline, you must first switch to Edit Mode. If you are not already in Edit Mode, switch back to Edit Mode now. Show the Quick Power Flow List and move it towards the bottom of the screen. Show bus 39820 by typing that number in the **Bus Number** field. Select **Network > Bus** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Click on the oneline towards the top center to define the point at which the new bus will be added. The Bus Information Dialog will appear. In the **Bus Number** field enter 39820. Select **Find by Number** to view the bus information. You should see the information appear in the Bus dialog fields that corresponds with the MAIN power flow case. Select *OK* to place the bus. The bus should now appear on your screen.

If it has not already been done for you automatically, add a bus field identifying the new bus' number immediately to the left of the new bus. To do this, click on or to the left of the display object that represents bus 39820 and select **Field > Bus Field** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. The Bus Field Options Dialog will appear for you to fill out. Designate the type of field as *Bus Number* and close the dialog. Simulator will add a text object showing the bus number at the point where you had clicked.

Repeat this procedure to place buses 39881 and 39821 on the oneline, along with their bus numbers.

---

**Tutorial: Automatic Line Insertion Page 7 of 15**

---

Transmission lines between buses can be inserted manually by choosing **Network > Transmission Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab and clicking on the beginning bus. Vertices may be defined along the way by clicking the mouse on the diagram where vertices should appear. However when creating a case from an existing power flow file, you also have the option to insert transmission line display objects automatically. To do this, select **Auto Insert > Lines** from the [Quick Insert](#) ribbon group on the [Draw](#) ribbon tab. Accept the default options and click OK. The lines joining the visible buses on the display are automatically added, along with circuit breakers and pie charts, provided those options are set. Simulator will draw only transmission lines that link buses that have already been drawn on the oneline diagram. If you add another bus to the diagram, you can again auto-insert lines, and Simulator will only insert lines that are not already present on the display.

---

**Panning and Zooming** Page 8 of 15

---


Two features of Simulator are indispensable when you have a large, detailed oneline: panning and zooming. To pan from side to side or up and down, either use the arrow keys or the scrollbars on the sides of the oneline. To zoom in or out of the oneline, hold the **Ctrl** key down and press the up arrow to zoom in and the down arrow to zoom out. For the example, pan up and insert buses 39819 and 39841, again with bus fields showing their numbers. Again use **Auto Insert > Lines** from the [Quick Insert](#) ribbon group on the [Draw](#) ribbon tab to add the transmission lines.



---

**Tutorial: Adding Background Page 9 of 15**

---


Sometimes you may wish to insert background elements on a oneline such as bodies of water or state lines to convey geographic location. To show static background elements on the oneline, select **Background > Background Line** from the [Individual Insert](#) ribbon group on the [Draw](#) ribbon tab. Click on the diagram to start the background line, and click at other points on the diagram to add segments. Double click to terminate the background line. If you wish to fill in the area inside the background line or change the background line's color, select the background line and choose **Line/Fill**  from the [Formatting](#) ribbon group on the [Draw](#) ribbon tab and make the appropriate choices in the resulting dialog. You may wish to experiment now with drawing background lines or objects and with adding fill color. Additionally, some political and geographic borders can be inserted by selecting **Auto Insert > Borders** from the [Quick Insert](#) ribbon group on the [Draw](#) ribbon tab.

---

**Tutorial: Simulating the Case Page 10 of 15**

---

Once you have constructed a oneline diagram showing the area of interest, you can simulate the case. Simulator does not only show the magnitude and direction of flows on transmission lines, but can also *animate* them. To configure the animated flows, switch to Edit Mode and choose **Online Display Options** from the [Active](#) ribbon group on the [Onelines](#) ribbon tab. This dialog box also allows you to change the size, density, and fill color of the animated flows for easier visualization. At this point, just click *OK*. Next, save your case by selecting **Save**

**Case** from the [Application Button](#) or by pressing the **Save Case**  button on the [Quick Access Toolbar](#). If you have not already saved the case, a **Save As** dialog will prompt you to select a name. Both the oneline diagram and the case will use this name.

To begin solving and animating the load flow solution, switch to Run Mode and then press the Play button in the Power Flow Tools ribbon group on the [Tools](#) ribbon tab. You should now see Simulator modeling the 3990-bus case.

---

**Tutorial: Run-time Object Dialogs Page 11 of 15**

---

While the animation ensues, you might want to view or change some study parameters. To do this, first pause the animation. (Note that you do not *have* to pause the animation to tweak parameters. You can adjust anything in the case while the animation runs, too.) Then, right-click on any of the objects on the display, and choose Show Dialog from the popup menu. This will bring up the run-time dialog for the object. Many of the parameters on this dialog can be modified, and the new settings will take effect when the animation is restarted.

---

**Online Local Menu** Page 12 of 15

---

Several options are available at run time from the oneline diagram's local menu. To call up the menu, right-click on an empty portion of the oneline. The local menu will appear. Use it to print the oneline, save it as a metafile, or copy it to the clipboard. You can also find a particular bus on the oneline, access panning and zooming options, set oneline display options, view information about the power system area in which you clicked, create a contour plot, and use the difference flows activity, all from the oneline local menu.

---

**Tutorial: Area Page 13 of 15**

---

Often, system data is most conveniently displayed by area. To view the Area Records Display, select **Aggregation > Areas** from the [Model Explorer](#). The resulting display summarizes information about all the areas in the case. You can sort the entries by clicking on the column labels.

---

**Limit Violations Page 14 of 15**

---

You can view a report of limit violations by selecting **Limit Monitoring Settings** from the [Other Tools](#) ribbon group on the [Tools](#) ribbon tab. A display showing bus voltage violations, line/transformer violations or interface violations will appear. To show only violations, select **Violating Elements** from the **Elements to Show** options. If the **Use Area / Zone Filters on List Displays** is checked, then the displayed violations will correspond only to areas whose **Shown** field is set to **Yes** on the [Area/Zone/Owner Filters](#) display.

If you wish to see more information on a bus that appears in the violation list, right-click on the bus number and select [Quick Power Flow List](#) from the resulting local menu. If the [Case Information Display](#) is set to refresh automatically, the list of bus voltage violations will update as new violations occur.

---

**Other Case Information Displays** Page 15 of 15

---

In addition to the few displays discussed in this tutorial, Simulator offers many other [Case Information Displays](#). In most cases, you can view information about buses, generators, lines, transformers, loads, and zones simply by right-clicking on the object in question and choosing the appropriate option from the object's local menu. The best way to become more familiar with the displays and the information contained in them is simply to play with a Simulator case and online. Simulator's interface has been designed to be simple and intuitive. If you run into problems, consult the On-line Help for assistance.

---

**Tutorial: Solving an OPF** Page 1 of 6

---

**Note: The OPF option in PowerWorld Simulator is only available if you have purchased the OPF add-on to the base package.** Please contact PowerWorld Corporation at [info@powerworld.com](mailto:info@powerworld.com) or visit the website at <http://www.powerworld.com> for details about ordering the OPF version of Simulator.

The PowerWorld Simulator (Simulator) is an interactive power system simulation package designed to simulate high voltage power system operation. In the standard mode, Simulator solves the power flow equations using a Newton-Raphson power flow algorithm. With the optimal power flow (OPF) enhancement, Simulator OPF can also solve many of the system control equations using an Optimal Power Flow algorithm. Specifically, Simulator OPF uses a linear programming (LP) OPF implementation.

The purpose of an OPF is to minimize an [objective \(or cost\) function](#) by changing different [system controls](#), taking into account both [equality and inequality constraints](#). These constraints are used to model the power balance constraints and various operating limits.

In Simulator OPF, the LP OPF determines the optimal solution by iterating between solving a standard power and then solving a linear program to change the system controls to remove any limit violations. See [OPF Primal LP](#) for more details.



---

**Tutorial: OPF Three Bus Example** Page 2 of 6
 

---

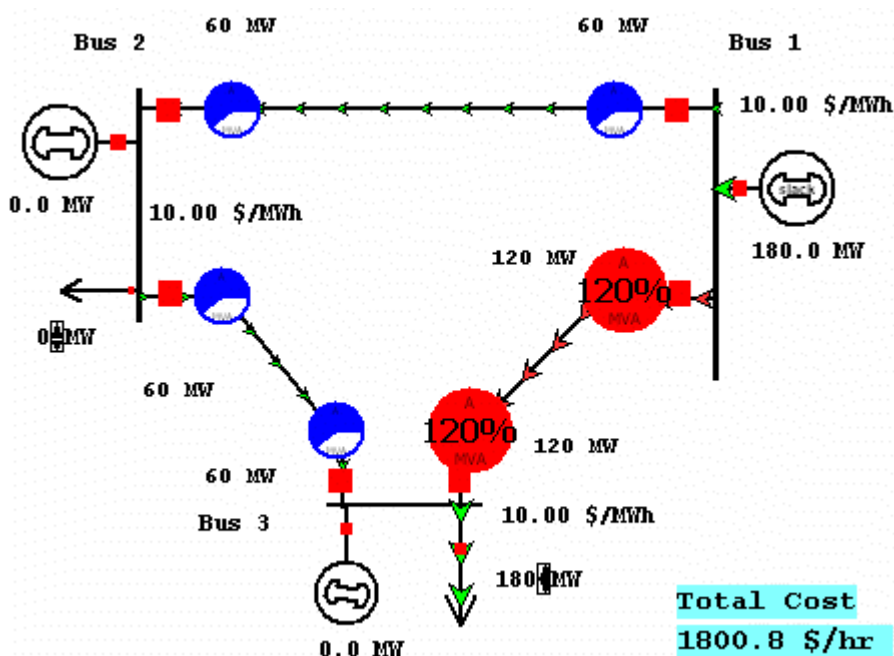
For this tutorial, we will start with a provided three-bus case (B3LP) found in the *PowerWorld\Simulator\Sample Cases* directory. Bus 1 is the system slack bus. All buses are connected via 0.1 pu reactance lines, each with a 100 MVA limit. There is a single 180 MW load at bus 3. The generator marginal costs are:

- Bus 1: 10 \$/MWhr; Range = 0 to 400 MW
- Bus 2: 12 \$/MWhr; Range = 0 to 400 MW
- Bus 3: 20 \$/MWhr; Range = 0 to 400 MW

To begin:

- Load B3LP case.
- Verify the above system specifications by right clicking on each line, bus, and generator and selecting its respective **Line/Bus/Generator Information Dialog** from the drop down menu.
- Select **Run Mode**.
- To be included in the OPF, all required area AGC status fields must be selected to OPF. To set area AGC to OPF: Select **OPF Case Info > OPF Areas** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab. The OPF Area Records display automatically opens. Verify the AGC Status field is set to **OPF**. If it is not, double click on the field to change its value. Close the OPF Area Records display.
- Select **Add Ons > Primal LP** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab to solve the case. Note: line limits are not initially enforced.

Your display should look similar to the following:



B3LP Solved Using Add Ons > Primal LP

Note the line from Bus 1 to Bus 3 is overloaded and that all buses have the same marginal cost.

---

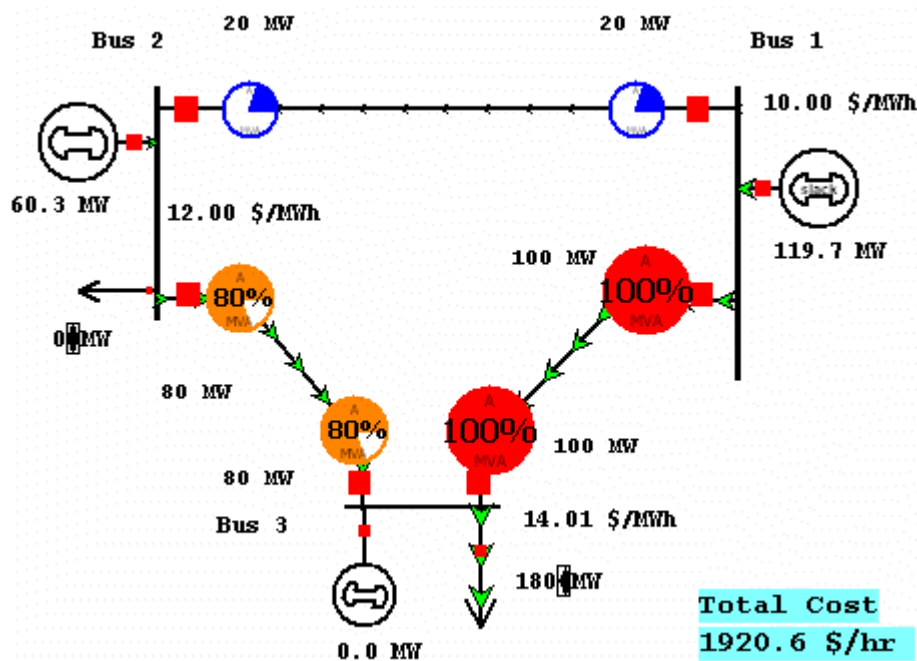
**Tutorial: OPF Line Limit Enforcement** Page 3 of 6
 

---

- Select **OPF Case Info > OPF Areas** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab.

The *Branch MVA* column specifies whether or not the MVA limits should be enforced for transmission lines and transformers that have at least one terminal in this area. For a transmission line or transformer to be included in the OPF constraints, *Line/Transformer constraints* must not be disabled on the **OPF Options and Results Dialog**, and the individual line/transformer must be enabled for enforcement on the **OPF Line/Transformer MVA Constraints** display.

- Double click on the *Branch MVA* field to change the value to **YES**. Close the OPF Area display
- Select **OPF Options and Results** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab.
- The **LP OPF Dialog** opens automatically. Select the **Constraint Options** tab then click *Disable Line/Transformer MVA Limit Enforcement* to remove the checkmark.
- Click **Solve LP OPF** then click **OK**.



B3LP Solved with Line Limit Enforcement Enabled

The LP OPF re-dispatches to remove the line limit violation. Bus marginal costs have changed from the initial example. The new bus marginal cost at Bus 3 is 14 \$/MWh. To verify this:

- Increase the load at Bus 3 by 1 MW. Right click on the load, enter "181.0" in the *Constant Power / MW Value* field in the **Load Options** dialog. Click **OK**.
- Select **Primal LP** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab.

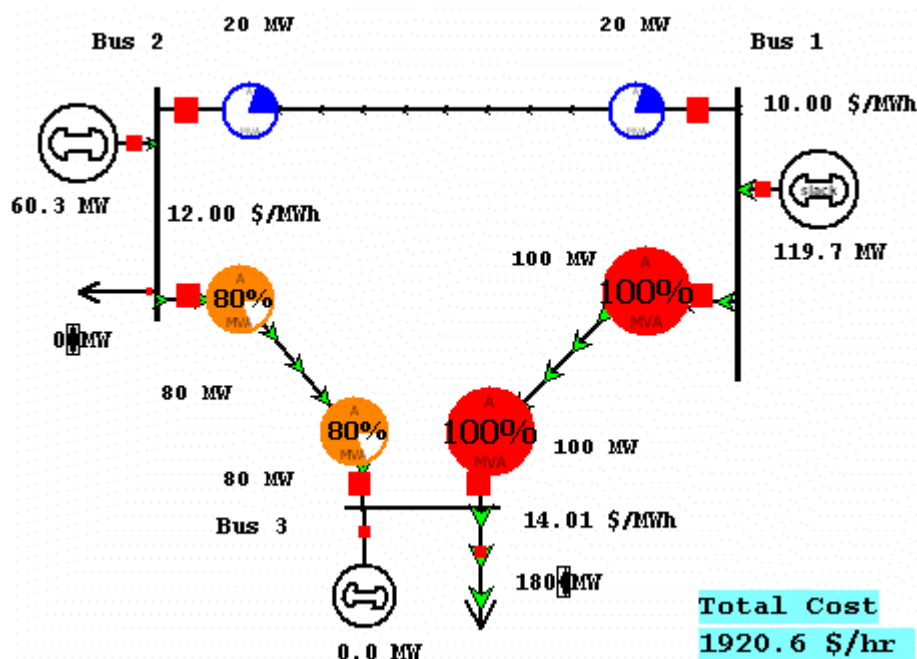
The Total Cost increased to 1935 \$/hr from the previous value of 1921 \$/hr, a difference of 14 \$/hr when Bus 3 load was increased by 1 MW.

---

**Tutorial: OPF LMP Explanation Page 4 of 6**


---

Explanation of Bus 3 LMP = 14 \$/MWh (from previous page):



B3LP Solved with Line Limit Enforcement Enabled

All lines have equal impedance. Power flow in a simple network distributes inversely to impedance of the path.

- For Bus 1 to supply 1 MW to Bus 3, 2/3 MW will take the direct path from 1 to 3, while 1/3 MW will take the path from 1 to 2 to 3.
- Likewise, for bus 2 to supply 1 MW to Bus 3, 2/3 will go from 2 to 3, while 1/3 will go from 2 to 1 to 3.
- To supply one additional MW to Bus 3, we need the change in power of generator 1 ( $P_{g1}$ ) plus the change in power of generator 2 to equal 1 MW.

$$P_{g1} + P_{g2} = 1 \text{ MW}$$

- With the line from 1 to 3 limited, no additional power flows are allowed on it.

$$(2/3)P_{g1} + (1/3)P_{g2} = 0$$

- Solving the above system of equations results in:

$$P_{g1} = -1 \text{ MW and } P_{g2} = 2 \text{ MW}$$

$$\Delta \text{cost} = \sum (\Delta P_{g_i} * \text{LMP}_i) = [(-1 \text{ MW}) * (10.00 \text{ $/MWh}) + (2 \text{ MW}) * (12.00 \text{ $/MWh})] = 14.00 \text{ $/hr}$$

---

**Tutorial: OPF Marginal Cost of Enforcing Constraints** Page 5 of 6
 

---

Similar to the bus marginal cost, you can also calculate the marginal cost of enforcing a line constraint. For a transmission line, this represents the amount of system savings that could be achieved if the MVA rating was increased by 1.0 MVA.

- Select **OPF Case Info > OPF Lines and Transformers** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab. The OPF Constraints Records dialog opens.

Note the column displaying **MVA Marginal Cost** displays **6.0** for the line from Bus 1 to Bus 3. This is determined based on the following:

- With no change in system load:

$$Pg1 + Pg2 = 0$$

- If we allow one additional MVA to flow on the line from Bus 1 to Bus 3:

$$(2/3)Pg1 + (1/3)Pg2 = 1$$

- Solving the above system of equations results in:

$$Pg1 = 3 \text{ MW and } Pg2 = -3 \text{ MW}$$

$$\Delta\text{cost} = \Sigma(\Delta Pg * LMP) = [(3 \text{ MW}) * (10.00 \text{ \$/MWh}) + (-3 \text{ MW}) * (12.00 \text{ \$/MWh})] = -6.00 \text{ \$/hr, a net savings of 6.00 \$/hr.}$$

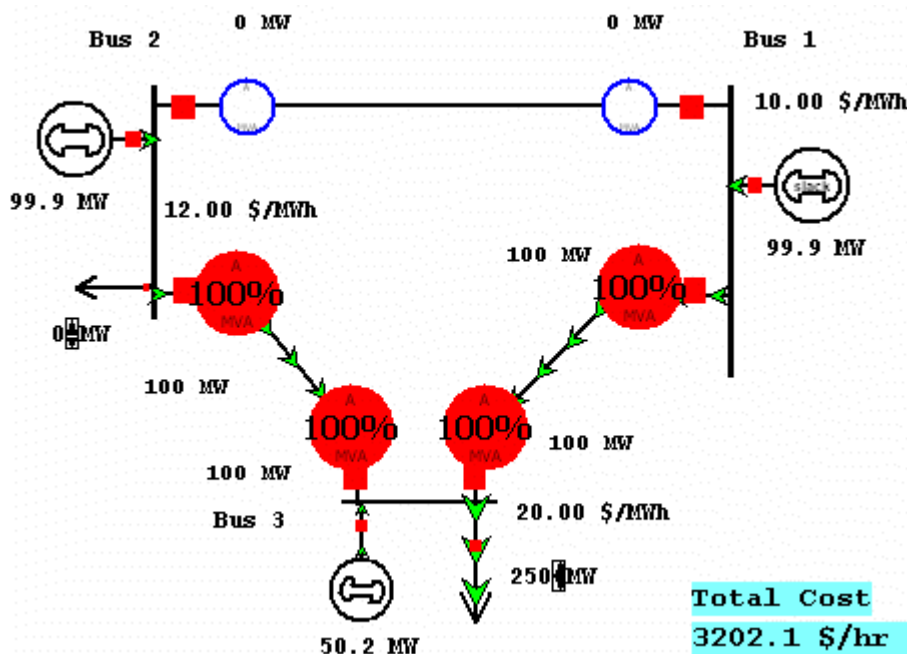
---

**Tutorial: OPF Unenforceable Constraints** Page 6 of 6
 

---

Next we will consider a case with unenforceable constraints. To begin, increase the load at Bus 3 to 250 MW then run the LP OPF again:

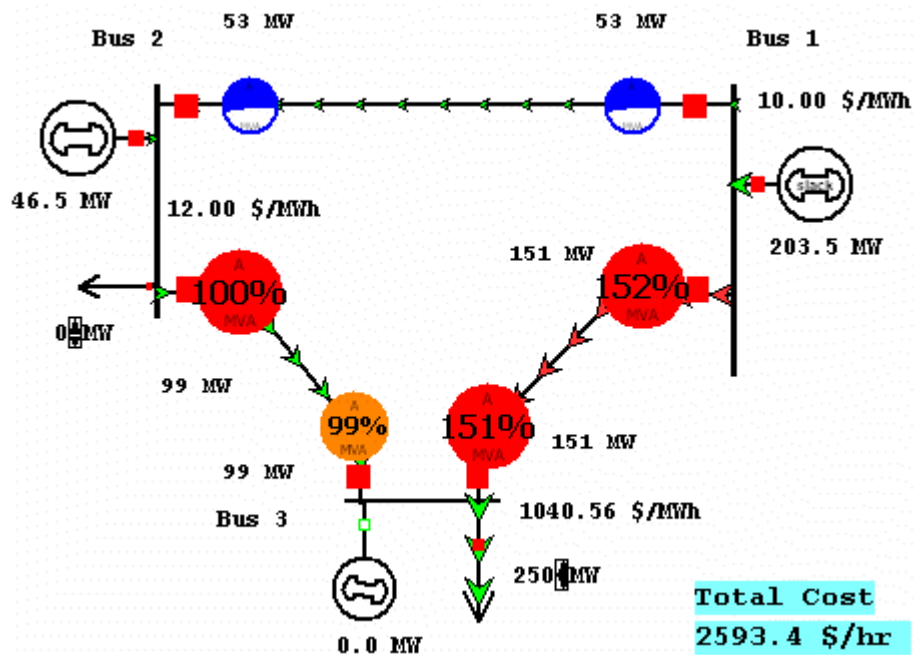
- Right-click on the load at Bus 3. Enter 250 in the Constant Power / MW Value field.
- Select **Primal LP** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab.



B3LP with Bus 3 Generator Carrying Load

Note that the transmission lines connecting Bus 3 to the other buses are both at their respective MVA limit and that the generator at Bus 3 is supplying the load in excess of the transmission line limits. Next we will open the generator at Bus 3 thereby inserting an unenforceable constraint:

- Left-click on the red breaker symbol connecting the generator to Bus 3 to open the breaker.
- Select **Primal LP LP** from the **OPF** ribbon group on the [Add Ons](#) ribbon tab.



#### B3LP Solved with Unenforceable Constraints

Both constraints cannot be enforced. If a constraint cannot be enforced due to insufficient controls, the slack variable associated with enforcing that constraint *can not* be removed from the LP basis.

Note the new LMP value for bus 3 exceeds 1000 \$/MWh. The marginal cost depends on the arbitrary cost of the slack variable. This value is specified in the *Marginal Violation Cost* field of the **Add Ons > OPF Options and Results** dialog. See [OPF Options and Results dialog](#) for more details.

#### Additional Example

To see another OPF example select **PowerWorld Simulator Add-on Tools > Optimal Power Flow (OPF) > Examples** from the table of contents in the on-line help file.

---

**Tutorial: Contingency Analysis**

---

This tutorial will walk you through the basic commands necessary to insert contingencies and have Simulator automatically analyze the results. Please see [Introduction to Contingency Analysis](#) for the necessary background information regarding the capabilities and uses of the Contingency Analysis tool.

For this tutorial, we will use an existing 7-bus case.

- Open case B7SCOPF from the *Program Files/PowerWorld/Simulator/Sample Cases* directory.
- Ensure Simulator is in [Run Mode](#).
- Select **Contingency Analysis** from the [Run Mode](#) ribbon group on the [Tools](#) ribbon tab. Simulator opens the Contingency Analysis Dialog.

The next section of the tutorial discusses Defining Contingencies and provides an example of inserting a single element contingency.

**1 2 3 4 5 6 7 8 9 10 11**

**Next**

# Index

## A

- Abort, 23, 49
- Absolute Value, 68
- ACE, 574, 575, 692, 701, 730
- Actions, 1148, 1161, 1167, 1174, 1178, 1180, 1182, 1183, 1186
- Add-On Tools, 916, 936, 990, 1049
- Admittance, 149, 639
- Advanced, 523
- Advanced Filter, 511, 522, 523
- Advanced Filters, 526, 536
- Advanced Sort, 503, 508
- AGC, 573, 575, 586, 660, 661, 669, 692, 728, 730
- Aggregation, 35
- Aggregations, 485
- Align Group Objects, 318
- Anchored, 33, 70, 157, 307, 317
- Anchored Objects, 157, 240, 317
- Anchors, 157, 240
- Animated Flows, 72, 81
- Animation, 63, 72, 81, 849
- Appending, 13, 239
  - Case, 13
- Appending a Case, 239
- Apply Template, 65
- Area, 3, 46, 167, 170, 200, 298, 304, 345, 430, 457, 463, 464, 465, 466, 467, 468, 521, 573, 575, 579, 702, 704, 705
  - Zoom, 46, 304
- Area Average Cost, 705
- Area Chart, 702
- Area Control, 692
- Area Control Error, 692, 701
- Area Display, 573, 575
- Area Display Objects, 159
- Area Display Options Dialog, 160
- Area Fields, 167, 170, 312, 468
- Area Information, 457, 459, 460, 461, 462, 463, 464, 465, 466, 467
  - Area Buses, 459
  - Area Gens, 460
  - Area Loads, 461
  - Area MW Control Options, 462
- Area Loss Sensitivity, 592
- Area Losses, 703
- Area MW Control Options, 462
- Area MW Transactions, 704
- Area Slack, 693
- Area Slack Bus Control, 693
- Area Transactions, 573, 575, 704
- Area/zone, 304
- Area/Zone/Owner, 521
- Area/Zone/Owner Filters, 26, 508, 511, 521
- Areas, 451, 457, 463, 464, 465, 521, 567, 573, 575, 610, 692, 701, 702, 703, 704, 705
- Associate, 1272
- ATC, 15, 853, 1016, 1017, 1031, 1033, 1039, 1180, 1197
  - Multiple Scenario, 1017, 1033
- ATC Analysis, 1033
- ATC Analysis Methods - Iterated Linear Step IL, 1044
- ATC Analysis Methods - Iterated Linear Step IL then Full CTG Solution, 1046
- ATC Analysis Methods - Single Linear Step SL, 1043
- ATC Analysis Methods - Solution Methods, 1042
- ATC Dialog Analysis Tab, 1028
- ATC Dialog Result Tab, 1030
- ATC Dialog Options Tab, 1019
- ATC Extra Monitors, 1031
- Augmenting Max Flow Min Cut, 293, 294



Auto Insert, 198, 270, 272, 274, 275, 276, 277, 278, 279, 741, 858, 860

- Borders, 279
- Contingencies, 741
- Display Objects, 198, 272, 274, 275, 276, 277, 278
- PTDF Directions, 858, 860

Auto Insert Contingencies, 811

Auto Insert Injection Groups, 643

Auto Size, 507

- Column Widths, 507

Auto Updating, 304

Auto Zoom, 304

Automatic Control, 375, 401, 438

Automatic Generation Control, 728

Automatic Line Tap Dialog, 252

Automatically Insert, 198, 270, 272, 274, 275, 276, 277, 278, 741, 860

- Borders, 279
- Contingencies, 741
- Display Objects, 198, 272, 274, 275, 276, 277, 278
- PTDF Directions, 858, 860

Automatically Open, 685

Autosize, 507

Auxiliary File

- Export Format Description, 547

Auxiliary Files, 138, 139, 140, 143, 144, 145, 744, 1108, 1142

- Formats, 139, 140, 143, 144, 145, 744

Available Contingency Actions, 732

Available Transfer Capability, 15, 853, 1016, 1017, 1033, 1039, 1197

Average Cost, 705

AVR, 375, 401, 438, 582, 586

## B

Back, 34, 308, 313

- Send, 34, 308, 313

Background, 34, 35, 68, 207, 208, 209, 210, 303, 304, 308, 313, 314

Color, 68, 303

Pictures, 210

Shapes, 207, 208, 209

Background Objects, 207, 208, 209, 210, 216, 217, 219, 312

Background\_Lines\_Dialog, 211

Base, 34, 308, 313, 399, 1041

- Level, 313

Base Case, 721, 722, 723, 724, 726, 1041

Base Case Violations, 772

Binding Constraint Dialog, 888

Bitmap format, 65

Blink, 69

Blocks, 748

BMP, 66

Branch, 68, 304, 610, 1039

- Object, 304

Branch Limiters, 1039

Branches, 572, 629

Branches that Create Islands, 712

Breakpoint, 482

Bring, 298, 313, 520

- Front, 34, 308, 313

Browse, 264

Browsing, 69

Build, 133, 134

Building New Case, 133

Bus, 3, 31, 33, 64, 68, 80, 157, 172, 173, 174, 241, 242, 244, 246, 247, 248, 257, 296, 304, 307, 319, 325, 328, 330, 418, 581, 582, 584, 706, 709, 782, 827

- Object, 304

Bus Field Information, 328

Bus Fields, 173, 312, 328

Bus Information, 393, 418

Bus Information Dialog, 507

Bus Load Throw Over Records, 782

Bus Mismatches, 584

Bus Options, 325

Bus Renumbering, 241, 242, 244

Bus View, 517, 706, 709

Buses, 46, 157, 172, 241, 246, 247, 248, 250, 270, 325, 330, 393, 418, 451, 567, 572, 634, 706, 709, 764, 827  
 Buyer, 1017, 1031

## C

Calculate Participation Factors, 730  
 Calculated Fields, 496  
 Cascade, 60, 115  
 Case, 13, 15, 64, 126, 129, 130, 132, 135, 239, 507, 566, 567  
     Appending, 13, 239  
     Submode, 15  
 Case Description, 566  
 Case Formats, 126, 566  
 Case Information, 13, 16, 488, 499, 503, 504, 507, 559, 566, 569, 571, 572, 573, 575, 584, 595, 620, 682  
 Case Information Customizations, 498  
 Case Information Displays, 16, 314, 488, 498, 499, 503, 504, 508, 521, 522, 523, 526, 527, 528, 529, 532, 534, 535, 539, 543, 544, 546, 559, 566, 573, 575, 581, 585, 595, 682  
     Custom Field Descriptions, 492, 494  
 Case Information Displays Colors and Cell Styles, 501  
 Case Information Displays Using Cell Handles, 502  
 Case Information Toolbar, 486, 507  
 Case Pathname, 568  
 Case References, 735  
 Case Summary, 567  
 Case Templates, 65  
 Case Totals, 568  
 Case Validation, 129  
 Center, 46, 109, 304  
     Display, 46  
     online, 304  
 Change, 303  
 Change button, 312  
 Change Field Data, 561

Change Fill Color, 312  
 Change Line Color button, 312  
 Change Timepoint Time Dialog, 897  
 ChangeParametersMultipleElement Function, 1061  
 ChangeParametersMultipleElement Sample Code, 1062  
 ChangeParametersMultipleElementFlatInput Function, 1064  
 ChangeParametersMultipleElementFlatInput Sample Code, 1065  
 ChangeParametersSingleElement Function, 1058  
 Charts, 701, 702, 703, 704, 705  
 Choosing, 313  
     Layer, 313  
 Circuit, 598  
 Circuit Breakers, 184  
 Close Online, 137  
 Closest Facilities to Point, 108  
 Color, 33, 68, 73, 301, 303, 307, 312, 842  
 Color Dialog, 303, 312  
     displays, 312  
 Color Key, 544  
 Color Map, 842  
 Colors, 309, 311, 312, 544  
 Column Metrics, 520  
 Column Widths, 507  
     Auto Size, 507  
 Comments, 17  
 Complete Show, 68  
 Conditional Display, 63, 117, 315  
 Conditional Display by Zoom Level, 315  
 Configuring, 508  
     Case Information Displays, 508  
 Connect, 1052  
 Context-sensitive, 9  
 Contingencies, 740  
 Contingency, 15, 733, 734, 735, 741, 744, 745, 747, 748, 749, 752, 753, 754, 757, 760, 762, 764, 766, 768, 769, 771, 774,

- 779, 782, 783, 785, 786, 787, 789, 792, 794, 805, 806, 807, 808, 1329
  - Blocks, 774
  - Case References, 735
  - Compare, 755
  - Delete, 754
  - Global Actions, 774
  - Records, 735, 747
  - Report, 754
  - Submode, 15, 1178
- Contingency Action, 733
- Contingency Actions, 754, 762, 766, 792
  - Compare Contingencies, 754
- Contingency Analysis, 3, 612, 731, 733, 734, 735, 740, 741, 744, 747, 748, 749, 752, 753, 754, 757, 760, 762, 764, 766, 768, 769, 772, 775, 777, 782, 783, 785, 786, 787, 789, 792, 794, 805, 806, 807, 808, 853, 869, 1017, 1329
  - Advanced Limit Monitoring, 771
  - Advanced Modeling, 774
  - Auto Insert Contingencies, 741
  - Compare Contingencies, 754
  - Load Contingencies, 744
  - Options, 768, 769
  - Reference State, 735
  - Saving Records, 747
  - Summary, 786
- Contingency Analysis Defining Contingencies, 740
- Contingency Analysis Definitions, 733
- Contingency Analysis Results, 755, 787
- Contingency Analysis Tutorial, 807, 808, 1329
- Contingency Blocks, 748, 749, 774
- Contingency Case References, 735, 738, 739
- Contingency Case References - State Information, 736
- Contingency Definition, 749, 754, 757, 762, 764, 766
- Contingency Element, 733
  - Contingency Element Dialog, 749, 762, 766, 794, 805
  - Contingency Global Actions, 774
  - Contingency Records Auto Insert, 811
  - Contingency Specific Options, 734
  - Contingency Violations, 754, 760, 764, 789
  - Contingency Violations Display, 757
  - Continuously Update, 544
- Contour, 65, 543, 544, 834, 835, 836, 840, 842
  - Contour Column, 520, 543, 544
  - Contour Type, 544, 836, 840
- Contouring, 499, 543, 544, 834, 835, 836, 840, 842
  - Contouring Options, 835, 840, 842
    - Color Key, 842
- Control Areas, 567
- Controls, 347, 695
- Converge, 16, 661, 691
- Convergence, 16
- Convergence Tolerance, 661
- Converters, 394
- Converting Background Ellipses, 214
- Converting Background Lines, 213
- Converting Background Rectangles, 215
- Coordinates, 109
  - Screen, 109
- Copy, 16, 31, 296, 319, 509, 538
  - Copy All, 509, 538
  - Copy All menu, 509
  - Copy Command, 31, 296, 319
  - Copy Online, 82
  - Copy Selection, 509, 538
  - Copy Selection menu, 509
  - Copy Window, 82
  - Copying, 82
  - Copying Simulator Data, 509, 538
- Cost, 141, 342, 346, 427, 588, 589, 591, 593, 705, 711
  - Cost Curve, 141, 711
  - Cost Information, 342, 346, 427

- Cost Models, 593
- Counter Flows, 849
- Create Equivalent, 282, 283
- Creating, 12, 1273
  - New Case, 12
  - New Project, 1273, 1274
- Creating Injection Groups, 642
- Criteria Dialog, 298
- Cross Compound Units, 142
- CTG, 733, 807, 808, 1329
- Current Zoom Level, 304, 315
- Curves, 349, 711
- Custom, 559, 842
- Custom Case Information Display, 559, 561
- Custom Case Information Displays, 498, 560, 561, 562, 564, 565
- Custom Color Map, 842
- Custom Detail Dialog, 68
- Custom Display, 561
- Custom Expressions, 518, 527, 528, 779
- Custom Expressions Display, 528
- Custom Hint Values, 713
- Custom Results Selection Dialog, 890
- Customize, 65, 67, 68, 72
- Cut, 31, 296, 319

## D

- Data LimitSet, 1228
- DC, 188, 392, 393, 394, 396, 567, 606, 607, 660, 671, 673
- DC Transmission Line, 389, 390
  - Actual Flow, 390
  - Inverter Parameters, 389
- DC Transmission Line Options, 385
- DC Transmission Lines, 386, 388
  - Line Parameters, 386
  - Rectifier Parameters, 388
- Decimal, 311, 507
- Decimal places, 507

- Decrease Decimals, 507
- Default, 254, 298, 303, 304, 306, 312, 313, 314, 315
  - Background Color, 303
  - Drawing Values, 306
- Default Drawing Options, 254
- Default Drawing Options Dialog, 31, 32, 296, 297, 306, 312, 319, 320
- Default Drawing Values, 306, 311
- Default Values, 305
- Define Expression, 518, 527
- Define Fields/Strings, 560
- Define Layers, 313
- Define Layers button, 313
- Defining, 740
- Defining Contingencies, 740
- Definitions, 733
- Delete, 31, 32, 296, 297, 314, 319, 320
  - Object, 32, 297, 320
- Deleting Injection Groups, 645
- Delphi, 1073, 1080, 1088
- Dialog, 298, 306
- Difference, 721, 723, 724
- Difference Case, 722, 723
- Difference Flows, 721, 722, 723, 724, 726
- Digits, 311
  - Right, 311
- Directions, 859, 860
- Disable Balancing, 661
- Disable Checking Gen VAR Limits, 661
- Disable Logging, 16
- Disabled
  - Why is this field or line disabled, 499
- Display, 32, 297, 315, 317, 320
  - Conditional, 315
  - Refresh, 508
- Display Detail, 68
- Display Grid Lines on Online, 301
- Display Log, 16
- Display Object, 31, 32, 33, 64, 68, 172, 176, 296, 297, 298, 307, 317, 319, 320

Display Object Properties, 317  
 Display Objects, 178, 180, 184, 188, 190, 193, 195, 197, 217, 249, 250  
 Display Options, 79, 156, 303  
 Display/Column Options, 507  
 Display/Size, 33, 307, 309, 317  
 Display/Size Tab, 309, 317  
 Displays, 299, 304, 312, 488, 706  
   Color Dialog, 312  
 Distance, 700  
 Document, 206  
 Document Links on Onelines, 206  
 Drawing Options, 254

## E

Economic Dispatch Control, 692  
 ED, 661, 692  
 Edit, 13, 31, 32, 152, 296, 297, 298, 301, 306, 313, 314, 315, 319, 320  
 Edit Menu, 31, 32, 296, 297, 319, 320  
 Edit Mode, 12, 13, 15, 31, 32, 33, 35, 65, 151, 152, 157, 296, 297, 306, 307, 313, 314, 315, 319, 320, 323, 325, 331, 335, 350, 357, 411  
 Edit Mode Introduction, 13  
 Edit Mode Overview, 13, 151  
 Edit Screen Layers, 314  
 Edit Toolbar, 31, 296, 319  
 Edit Tools, 31, 296, 319  
 Enter new keyword here, 607  
 Environment, 659, 677  
 Equipment Mover, 249, 250  
 Equivalencing, 282, 283  
 Equivalencing Display, 257  
 Equivalentents, 257, 282  
 Equivalentents Display, 283  
 Excel, 509, 538, 1106  
 Export, 622, 697  
 Extra Monitors, 1031  
 Extra Width, 301

## F

F1 key, 9  
 Facility Analysis, 257, 290, 291, 293  
 Fault, 15, 344, 824, 828, 830, 833  
   Submode, 15, 1182  
 Fault Analysis, 823, 824, 827, 828, 829, 832, 833  
 Field, 35, 298, 311, 328, 540  
 Field Options, 468  
 Fields, 86, 157, 158, 168, 170, 173, 179, 181, 183, 192, 194, 196, 199, 216, 219, 261, 328, 380, 382, 383, 407, 416, 540  
   Key Fields, 540  
 File, 125, 568, 1270  
 File Management Options, 686  
 File Menu, 125  
 File options, 16  
   Save Log, 16  
 Fill Color, 309, 312  
 Fill Options, 312  
 Filter, 26, 508, 511, 521, 522, 523, 526, 536  
 Filters, 261, 521, 522, 536, 667  
 Find, 304, 504, 507, 532  
 Find Branches that Create Islands, 712  
 Find Dialog, 532  
 Find Records, 504  
 Finding Records, 504, 532  
 Fixed Screen Size, 313  
 Flow Sensitivities, 861  
 Flow Visualization, 72  
 Flowgates, 202, 203  
 Flows, 67, 68, 72, 200, 861  
 Font, 16, 33, 307, 309, 311, 313  
   choosing, 311  
 Font Properties, 311  
 Font Tab, 33, 307, 309, 311  
 Form Control, 499  
 Format, 33, 117, 126, 138, 299, 307, 309, 311, 313, 314, 315, 317, 507, 745  
 Format Field Properties, 316

- Format Menu, 314
- Format Multiple Objects, 117, 309
- Format Multiple Objects dialog, 313, 315, 317
  - Levels/Layers Tab, 315
- Format Multiple Options dialog, 309
  - open, 309
- Format Selection Dialog, 299, 309
  - Font Tab, 33, 307
  - Levels/Layers Tab, 33, 307
  - Line/Fill Tab, 33, 307
- Format Toolbar, 33, 307
- Front, 34, 308, 313
  - Bring, 313
- Fuel Cost, 591
- Fuel Type, 591
- Full CTG Solution, 1016
- Functional Description of Contour Options, 844
- Functions, 529
- Functions and Operators, 527, 528, 529

## G

- Gauges, 35
- gcd, 140
- gcp, 139
- General, 674, 1148
- General Script Actions, 15, 1148
- Generation Chart, 702
- Generation Shift Factor, 871
- Generation Shift Factor Sensitivities, 871
- Generator, 68, 142, 250, 335, 337, 339, 340, 342, 344, 345, 346, 347, 349, 423, 425, 427, 429, 430, 586, 588, 589, 591, 593, 685, 695, 711, 828
  - Limits, 685
  - MW Limits, 685
  - Ramp Limits, 685
- Generator Capability Curves Format, 139
- Generator Cost, 586, 589
- Generator Cost Curve Data Options, 141

- Generator Cost Data Format, 140
- Generator Costs, 593
- Generator Cubic Cost, 591
- Generator Display Objects, 178
- Generator Economic Curves, 711
- Generator Field Information, 337
- Generator Fields, 179, 312, 337
- Generator Information, 339, 342, 345, 346, 423, 425, 427, 429, 430
  - Area, 345, 430
  - Costs, 342, 427
  - Owner, 345, 430
  - Zone, 345, 430
- Generator MW Marginal Cost, 591
- Generator Options, 335, 340, 344
  - Fault Parameters, 344
  - MW Control, 340
- Generator Piecewise Linear, 593
- Generator Reactive Power Capability Curve, 349
- Generator Records, 586, 828
- Generator Spinning Reserves, 568
- Generator/Load Cost Models, 588
- Generators, 79, 157, 178, 179, 249, 250, 274, 286, 337, 339, 346, 347, 349, 423, 426, 429, 567, 572, 582, 586, 588, 589, 591, 593, 613, 615, 617, 636, 641, 642, 645, 646, 648, 651, 652, 655, 685, 695, 711, 805, 828
  - Limits, 685
- Generic Model Fields, 219
- Geographic Data View, 550
  - Geographic Data View Styles, 557
  - Fields and Attributes, 552
  - General Display Options, 556
- Geography/Coordinates, 95
- Get Column Metrics, 520
- Get Parameters, 1071, 1073, 1075, 1076, 1078, 1080, 1081, 1082, 1085, 1131
- GetParametersMultipleElementFlatOutput Function, 1084
- Getting, 12

- Started, 12
- GIS, 92, 93, 94, 96, 103, 105, 106, 108
  - Great Circle Distance Calculation, 93
  - Map Projections, 88
- GIS Shapefile Data, 97, 99, 100, 101
  - Control, 97
  - Format, 100
  - Identify, 98
  - Modify, 99
  - Shape List, 101
- GIS Tools, 92, 93, 94, 103, 105, 106, 108
  - Closest Facilities to Point, 108
  - Delete All Measure Lines, 106
  - Export Oneline as Shapefile, 103
  - Great Circle Distance, 92
  - Insert Measure Line, 105
  - Populate Lon,Lat with Display X,Y, 94
  - Shape File Import, 96
- Global, 748
- Global Actions, 748, 774
- Governor, 729, 730
- Graph, 294, 701, 702, 703, 704
- Graphic Formats, 147
- Graphics, 148, 505
- Great Circle Distance Calculation, 93
- Great Circle Distance Dialog, 92
- Grid, 301, 520, 546
  - Snap Objects, 301
- Grid Metrics, 546
- Grid Metrics Dialog, 520
- Grid/Highlight Unlinked Objects, 67, 301
- Gridline Color, 301
- Gridlines, 301
- GSF, 871

## H

- Help, 9
- High Zoom Level, 315
- Highlight Unlinked Objects, 301
- Highlighting, 301

- Unlinked Objects, 301
- Hourly Summary, 880, 882
- Local Menu, 882

## I

- Identify Cross Compound Units, 142
- Images, 148, 505
- Immobile, 33, 307, 317
- Impedance, 362, 399, 400, 444, 831
- Impedance Correction Tables, 444
- Import, 622, 697
- Import PTI Subsystems Dialog, 650
- Include, 1051
- Include Suffix, 311
- Increase Decimals, 507
- Increase/Decrease Decimals, 507
- Indication, 35
- inf, 144
- Info/Interchange, 463
- Initialize, 1271
- inj, 143
- Injection Groups, 68, 143, 568, 613, 615, 618, 641, 642, 645, 646, 648, 650, 651, 653, 655, 1225
  - Import PTI Subsystems Dialog, 650
- Injection Groups Format, 143
- Input Page, 883
- Insert, 35, 158, 198, 267, 270, 272, 274, 275, 276, 277, 278, 314, 481
  - Area, 35
  - Area/Zone, 35
  - Background, 35
  - Directions, 860
  - Display Objects, 35
  - Fields, 35
  - Hyperlink, 35
  - Injection Group, 35
  - Interface, 35
  - Link, 35
  - Memo, 35

Picture, 35  
 Pie Charts, 35  
 Screen Layer, 314  
 Text, 35  
 Zone, 35  
 Insert Palettes, 267  
 Install, 1050  
 Installing, 1050  
 InterArea, 200  
 Interchange, 573, 575  
 Interchange MW, 573, 575  
 Interface Data Format, 144  
 Interface Fields, 199, 479  
 Interface Information, 476, 478, 479, 480  
 Interface Limiters, 1039  
 Interfaces, 68, 144, 197, 198, 199, 201, 202, 277, 304, 451, 476, 478, 479, 480, 481, 482, 568, 610, 612, 764, 850, 1039, 1226  
 Introduction, 4, 915  
 Inverter Parameters, 389  
 IOA, 591  
 Island Based AGC, 669  
 Island Slack Bus, 620  
 Island-Based AGC, 660  
 Islands, 568, 620, 669, 728, 730  
 Iterated Linear Step, 1016, 1040  
 Iterations, 16, 661, 691

## J

Jacobian, 149, 638

## K

Key Field, 683  
 Key Fields, 540  
 Keyboard Short Cut Actions Dialog, 118  
 Keyboard Shortcuts, 118

## L

Labels, 451, 455  
 1338

Layer, 33, 298, 307, 309, 313, 314, 315  
     Screen Stack, 34, 308  
     Stack, 33, 307  
 Layers, 112, 313, 315  
     choosing, 313  
 Level, 33, 112, 298, 307, 312, 313, 315  
     Background, 34, 308  
     Base, 34, 308  
     Middle, 34, 308, 313  
     Top, 34, 308, 313  
 Levels/Layers, 33, 307, 309, 313, 314, 315  
 Levels/Layers Options, 313, 314  
 Levels/Layers Tab, 33, 307, 309, 313, 315  
     Format Multiple Objects dialog, 315  
 Limit, 685, 1016  
 Limit Groups, 629, 685  
 Limit Monitoring, 768, 769, 771  
 Limit Monitoring Settings, 624, 625, 629, 685, 1016, 1041  
 Limit Monitoring Settings and Limit Violations Dialog, 625  
 Limit Violations, 685  
 Limiters, 1039  
 Line, 31, 32, 33, 68, 252, 272, 296, 297, 298, 307, 309, 312, 319, 320, 598, 600, 637, 829  
 Line Color, 312  
 Line Field Options, 380  
 Line Fields, 183, 312  
 Line Flow Gauge Options Dialog, 187  
 Line Flow Gauges, 186  
 Line Information, 378, 396  
 Line Loading Replicator, 715  
 Line Outage Distribution Factors, 853  
 Line Per Unit Impedance Calculator Dialog, 362  
 Line Shunt Display, 605  
 Line Shunts, 361  
 Line Shunts Information, 361  
 Line Taps, 251, 252  
 Line Thickness, 312



Line/Fill, 33, 307, 309, 312  
 Line/Fill Properties, 312  
 Line/Fill Tab, 33, 307, 309, 312  
 Line/Transformer Information, 434  
 Linear ATC, 1017, 1040  
 Linear Calculation Method, 1017  
 Linearized AC, 1017  
 Lines, 764  
 Lines Buses and Interfaces Tab -  
 Contingencies Section, 765  
 Lines/Transformers, 567, 829  
 Links, 204, 206  
     Other Onelines, 204  
 List of Devices, 1086, 1088, 1089, 1090  
 ListOfDevicesAsVariantStrings Function,  
 1092  
 ListOfDevicesFlatOutput Function, 1093  
 Load, 80, 350, 354, 356, 431, 588, 595,  
 782, 805, 832  
 Load Benefit Models, 596  
 Load Chart, 702  
 Load Display Objects, 180  
 Load Field Information, 354  
 Load Fields, 181, 312, 354, 782  
 Load Flow, 17  
 Load ID, 595  
 Load Information, 431  
 Load Modeling, 356  
 Load Multiplier, 573, 575  
 Load NERC Flowgates, 202  
 Load Options, 350, 352, 353  
     Load Information, 352  
     OPF Load Dispatch, 353  
 Load Schedules, 573, 575  
 Load State, 1094, 1095  
 Load Status, 595  
 Load Throw Over, 782  
 Load Variation, 573, 575  
 Loading NERC Flowgates, 202  
 Loads, 157, 180, 181, 249, 250, 275, 286,  
 350, 354, 431, 451, 567, 572, 588, 595,

596, 613, 615, 617, 635, 641, 642, 645,  
 646, 648, 651, 652, 655, 805, 832  
 Local Menu, 65, 499  
 Local Menus, 565  
 LODF, 853, 855, 857, 1016  
 Log, 16, 23, 49  
     Windows, 16  
 Loss, 863  
     Sensitivities, 863  
 Loss Sensitivities, 863  
 Loss Sensitivity, 592  
 Losses, 673, 703  
 LTC Transformer, 375, 401, 438  
 LTC Transformers, 403, 440

## M

Maintain Fixed Screen Location, 313  
 Maintain Fixed Screen Size, 313  
 Make Default, 311  
 Make Default button, 311  
 Make-Up Power Sources, 805, 806  
 Map Projections, 88  
 Matrix Grids, 884  
 Max Flow Min Cut, 293, 294  
 Memo, 17, 467  
 Memo Display, 17  
 Memo Text, 218  
 Menus, 125  
 Merging Buses, 246  
 Message Log, 16, 23, 49, 691  
     Print, 16  
     Save, 16  
     Show, 16  
     Using, 16  
 Metrics, 520, 546  
 Microsoft Excel, 509  
 Middle, 34, 308, 313  
     Level, 313  
 Minimal, 68  
 Minimal Show, 68

Minimum Highlighted Object Pixel Size, 301

Minimum Sensitivity, 661

Mismatch, 584

Model Conditions, 667, 1231

Model Conditions Dialog, 535

Model Conditions Display, 535

Model Criteria, 667

Model Explorer, 485

    Explore Pane, 485

    Options Pane, 486

    Recent Tabs, 486

Model Expressions, 777

Model Fields, 219

Model Filters, 536, 667, 1233

Model Object, 64

Modeling, 356, 535, 536, 588, 589, 593, 596, 728, 768, 774

Models, 535, 589, 667

Moderate Show, 68

Modify, 12, 309, 312, 313, 314, 315, 317

    Orientation, 317

    Size, 317

Mouse, 3, 11, 68

Mouse Wheel Zooming, 68

Movie Maker, 87

Multiple Display Fields, 158

Multiple Scenario, 1039

Multiple Scenario Analysis, 1017, 1033

Multiple Scenario ATC Analysis - Results Tab   Local Menu Options, 1038

Multiple Scenario ATC Dialog   Results Tab, 1036

Multiple Scenario ATC Dialog   Scenarios Tab, 1034

Multi-Section Lines, 189, 378, 572, 600, 1238

Multi-Terminal, 392, 393, 394, 396, 607

Mutual, 830, 831

Mutual Impedance, 830, 831

MVA flow, 70

Mvar, 68, 403, 440

1340

Mvar Control, 403, 440

MVR flow, 70

MW, 68, 70, 699, 704

MW Control, 340

MW Distance, 699

MW Marginal Cost, 591

MW Transactions, 573, 575, 621, 704

MW/MVR, 68

## N

NERC, 202, 203

Network, 35, 485

Network Cut, 261, 262

New, 133, 134

New Case, 12, 13, 133

    Creating, 12

New Online, 13, 134

New Timepoint Dialog, 896

Newton, 56, 57, 688, 689

Newton-Raphson, 663, 691

Nomogram Display, 612

Nomogram Information, 482

Nomogram Interface Limiters, 1039

Nomograms, 482, 612

## O

O&M, 591

Objects, 17, 31, 32, 35, 64, 67, 68, 249, 259, 296, 297, 298, 304, 306, 309, 311, 312, 313, 314, 315, 317, 319, 320, 507, 1119

    Area, 304

    Area/Zone, 304

    Background Objects, 207, 208, 209, 210, 216, 217, 219, 312

    Bus, 304

    Display, 31, 32, 296, 297, 319, 320

    Find, 304

    Zone, 304

Observing, 313

- online, 313
- Off-Nominal, 400
- Online, 31, 33, 35, 63, 64, 67, 68, 77, 79, 81, 109, 110, 117, 120, 131, 136, 137, 147, 156, 167, 168, 204, 264, 296, 298, 301, 303, 304, 307, 311, 313, 314, 315, 317, 319, 679, 706, 709
  - Browsing Path, 680
  - Conditional Display, 110, 117
  - Coordinates, 680
  - Diagrams, 63, 68, 120, 131, 156, 167, 168, 170, 172, 173, 179, 181, 184, 185, 255, 303
  - Display, 67, 68, 79, 156, 301, 303
  - Display Options, 679
  - Displays, 706, 709
  - Exporting, 147
  - File, 679
  - Font, 680
  - Links, 69, 204
  - Options, 68, 79, 117, 156, 303, 313, 315, 679
  - Save, 680
  - Screen Coordinates, 109
  - Size, 680
  - Text Fields, 216
  - Visualizing, 679
- Online Display Options, 67, 95
  - Geography/Coordinates, 95
- Open, 16, 130, 131, 132, 298, 309, 313, 1276
  - Project, 1276
  - Screen Layers list, 313
- Open Case, 1096, 1097
- Operators, 529
- OPF, 353, 429, 466, 612, 693, 731, 936
  - Constraints, 938, 939, 940, 942, 943, 950
  - Controls, 937, 948, 952, 964, 980, 981
  - Data, 944, 947, 956
  - Example, 946, 982, 985, 986, 988
  - Load Dispatch Options, 353
  - LP OPF, 945, 959, 960, 961
  - Options, 947, 948, 950, 952, 956, 959, 960, 961, 962, 963, 964, 965
  - Records, 944, 966, 968, 969, 972, 973, 974, 976, 978
- OPF Control, 693
- OPF DC Lines Records, 975
- OPF Options Advanced Options, 955
- OPF Options and Results, 947
- Optimal Power Flow, 429, 466, 612, 693, 731, 936
  - Constraints, 938, 939, 940, 942, 943, 950
  - Controls, 937, 948, 952, 964, 980, 981
  - Data, 944, 947, 956
  - Example, 946, 982, 985, 986, 988
  - LP OPF, 945, 959, 960, 961
  - Options, 947, 948, 950, 952, 956, 959, 960, 961, 962, 963, 964, 965
  - Records, 944, 966, 968, 969, 972, 973, 974, 976, 978
- Options, 16, 68, 156, 254, 298, 301, 303, 304, 306, 309, 312, 314, 315, 507, 659, 660, 661, 669, 671, 674, 675, 677, 682, 691, 700, 728
- Options/Tools, 67, 81, 303, 306, 701, 726, 861
- Orientation, 309, 317
- Orientation Setting, 317
- OTDF, 1031
- Other Applications, 538
- Other Display Object Properties, 317
- Out of Service Elements, 679
- Outages, 572
- Out-of-Service Elements, 69
  - Visualizing, 69
- Overheated Lines, 685
- Overview, 63, 151, 266, 290, 641, 651, 916, 936, 990, 1270
- Owned, 635, 636, 637
- Owned Bus Records, 634
- Owned Generator Records, 636
- Owned Line Records, 637

- Owned Load Records, 635
- Owner, 345, 430, 521, 632, 633, 634, 635, 636, 637, 1240
- Owner Display Objects, 165
- Owner Display Options Dialog, 166
- Owner Fields on Onelines, 171
- Owners, 521

## P

- Palettes, 266, 267
- Pan, 3, 46, 110, 304, 313
  - Zoom, 110
- Pan/Zoom, 110, 304
  - New Location, 305
- Pan/Zoom Control, 111, 304
- Panning, 304
- Participation Factor Control, 692
- Participation Factors, 347, 587, 692, 695, 730
- Participation Points, 617, 651, 652, 655
- Paste, 31, 32, 296, 297, 319, 320, 509, 510, 538
- Paste Buffer, 31, 32, 296, 297, 319, 320
- Paste Format Dialog, 310
- Path, 69, 330
  - Browsing, 69
- Pathname, 568
- Pause, 23, 49, 87
- Per Unit, 362, 399
- Phase, 400, 405, 442, 602
- Phase Shift, 400, 405, 442
- Pie Chart / Gauge Dialogs, 225
- Pie Chart / Gauge Example, 237
- Pie Chart / Gauge Style Dialog, 229
- Pie Chart / Gauge Style Dialog - Gauge Parameters Tab, 235
- Pie Chart / Gauge Style Dialog - Open Parameters Tab, 233
- Pie Chart / Gauge Style Dialog - Pie Chart Parameters Tab, 234

- Pie Chart / Gauge Style Dialog - Standard Parameters Tab, 231
- Pie Chart Relative, 70
- Pie Chart Style, 70
- Pie Charts, 67, 70, 185, 201, 480
- Pie Charts/Gauges General Options, 224
- Pie Charts/Gauges Interfaces, 222
- Pie Charts/Gauges Lines, 220
- Pies, 35
- Populate Lon,Lat with Display X,Y, 94
- Popup menu, 314
- Post Power Flow, 661, 667
- Post-contingency, 1041
- Potential Misplacements Dialog, 250
- Power Control, 425
- Power Flow, 16, 569, 571, 584, 660, 661, 673, 691, 730, 782, 848
  - Submode, 1174
  - Visualize, 848
- Power Flow List, 569, 571
- Power Flow Solution, 3, 16, 622, 659, 660, 661, 667, 669, 671, 673, 674, 675, 677, 679, 684, 685, 691, 692, 697, 699, 700, 701, 702, 703, 704, 705, 706, 709, 728, 730, 782, 853
- Power Flow Solution Advanced Options, 664
- Power Flow Solution Message Log, 676
- Power Flow Solution Actions, 661
- Power System Model, 31, 32, 64, 296, 297, 319, 320, 356, 451, 535, 536, 588, 589, 593, 596
- Power System Scaling, 286
- Power Technologies, 744
- Power Technology, 745
- Power Transfer Distribution Factors, 848, 849, 1016
- PowerFlow, 15
  - Submode, 15, 1174
- PowerPoint, 82
- PowerWorld, 9, 12, 1119
- PowerWorld Corporation, 9

PowerWorld Project, 1270, 1271, 1272, 1273, 1274, 1276

PowerWorld Simulator, 9, 12

Present Case, 721, 722, 723, 724, 726

Present Topological Differences, 726

Presentations, 82

Pre-Transfer Value Estimate, 1041

Prevent Controller Oscillations, 662

Print, 120, 121, 123

Print Message Log, 16

Print Options, 121

Print Title Bar, 121

Print Window, 121

Printer Setup, 123

Printing, 120, 121, 123

ProcessAuxFile, 1098, 1099

Project, 1270, 1271, 1272, 1273, 1274, 1276

Properties, 301, 311, 312, 317

Display Object, 317

PSS/E, 744, 745, 747

PSS/E Load Throw Over Files, 746

PTDF, 3, 641, 699, 848, 849, 853, 858, 859, 860, 866, 1016, 1031, 1041

Directions, 859

Publish, 148, 505

PV, 15, 916, 917, 918, 919, 926, 928, 1183

Options, 920, 924, 935

Submode, 15, 1183

PVQV, 916, 917, 918, 919, 920, 923, 925, 926, 928, 929, 931, 932, 933, 934, 935, 1183

Options, 920, 924, 929, 933, 935

## Q

Quick Access Toolbar, 22

Quick Power Flow List, 517, 537, 569, 571

QV, 15, 919, 929, 931, 932, 933, 934, 1186

Options, 920, 924, 929, 933, 935

Submode, 15, 1186

## R

Range, 537

Range of Numbers, 537

Reactive Power Capability, 349

Recently Opened Cases, 132

Records, 64, 503, 504, 509

Sorting, 499, 503, 508

Rectangle Selector, 309

Rectangular Selector, 309

Rectifier Parameters, 388

Reference State, 735

Refresh, 59, 68, 114, 240, 508

Refresh Anchors, 240

Refresh Display, 508

Regulation, 587

Relationship Between Display Objects, 31, 32, 296, 297, 319, 320

Relinking, 301

Remote Regulation, 582, 586

Remotely Regulated, 582, 586

Renumber Areas/Zones/Substations Dialog, 245

Renumbering, 241, 242, 244

Report Violations, 771

Report Writing, 768, 783

RequestBuildDate Property, 1118

Required Field, 683

Required Fields, 542

Resize, 33, 307, 309, 313

Resizing, 313

Restore, 675

Restore Default Values, 305

Restore Last Solution, 675

Restore State, 675

Result Tab, 1039

Results, 885

Constraints, 885

Results Grid Pages, 891

## Ribbon, 19

Case Information, 26

Draw, 29, 154

Clipboard, 31, 296, 319

Formatting, 310

Individual Insert, 35

Quick Insert, 37

Select, 39, 300

## Onelines

Active, 42

Onelines, 40

Onelines

Zoom, 46

Options, 47

Quick Access Toolbar, 22

Tools, 49

Tools

Edit Mode, 51

Tools

Other Tools, 52

Tools

Run Mode, 54

Window, 59, 114

Right-clicking, 314

online, 314

Run, 14, 16

Run Mode, 14, 15, 69, 418, 421, 423, 431, 434, 445, 449, 848, 1167

Contouring, 834, 835, 836, 840, 842

Difference Flows, 721, 722, 723, 724, 726

Fault Analysis, 823, 824, 827, 828, 829, 832, 833

Governor Power Flow, 728, 729, 730

Introduction, 14

LODF, 853, 855

Options, 729

Script, 1167

Sensitivities, 861, 863, 871

Tools, 711, 721, 728, 731, 748, 777, 805, 834, 848, 853, 866, 871

Tools & Options, 87, 711, 721, 722, 723, 724, 726, 728, 730, 731, 734, 735, 741, 744, 745, 747, 748, 749, 752, 753, 754, 757, 760, 762, 764, 766, 768, 769, 771, 774, 777, 782, 783, 785, 786, 787, 789, 792, 794, 805, 806, 823, 824, 827, 828, 829, 830, 831, 832, 833, 834, 835, 836, 840, 842, 848, 849, 853, 855, 857, 858, 859, 860, 861, 863, 866, 867, 869, 871

RunScriptCommand, 1100, 1101

## S

Save, 16, 46, 112, 135, 136, 149, 539, 1102, 1104

Save Case, 539, 1102

Save Case Information Data, 539

Save Hidden Layers, 113

Save Log, 16

Save View, 46, 112

Save/Edit/Delete, 112

Saving NERC Flowgates, 203

Scaling, 286, 537

Schedules, 573, 575

SCOPF, 429, 466, 612, 692, 731, 990, 991, 992, 993, 994, 999, 1002, 1004, 1005, 1009

Constraints, 995, 996, 997

Controls, 999, 1010

Example, 1011, 1014, 1015

LP, 1004, 1007, 1008

LP OPF, 945, 959, 960, 961

OPF Constraints, 938, 939, 940, 942, 943, 950

OPF Controls, 937, 948, 952, 964, 980, 981

OPF Data, 944, 947, 956

OPF Example, 946, 982, 985, 986, 988

OPF Options, 947, 948, 950, 952, 956, 959, 960, 961, 962, 963, 964, 965

OPF Records, 944, 966, 968, 969, 972, 973, 974, 976, 978

Screen Coordinates, 109

Screen Layer Options, 315

- Screen Layer Options dialog, 314
- Screen Layers, 314, 315
- Screen Layers list, 313
  - open, 313
- Screen Location, 304, 313
- Screen Size, 313
  - Fixed, 313
- Script Command Execution Dialog, 15
- Script Command Execution Window, 15
- Script Commands, 15
- Script General Actions, 15, 1148
- Script Mode, 15, 23, 49, 1142, 1144, 1146, 1188, 1192
  - Actions, 1148, 1161, 1167, 1174, 1178, 1180, 1182, 1183, 1186
  - Data, 1199, 1200
  - Data, 1188, 1189, 1192, 1193, 1194, 1195, 1196, 1197
  - Data, 1201
  - Data, 1202
  - Data, 1203
  - Data, 1204
  - Data, 1205
  - Data, 1211
  - Data, 1212
  - Data, 1213
  - Data, 1214
  - Data, 1215
  - Data, 1216
  - Data, 1218
  - Data, 1220
  - Data, 1221
  - Data, 1223
  - Data, 1224
  - Data, 1225
  - Data, 1226
  - Data, 1228
  - Data, 1229
  - Data, 1230
  - Data, 1231
  - Data, 1232
  - Data, 1233
  - Data, 1234
  - Data, 1238
  - Data, 1239
  - Data, 1240
  - Data, 1243
  - Data, 1244
  - Data, 1245
  - Data, 1246
  - Data, 1247
  - Data, 1248
  - Data, 1249
  - Data, 1250
  - Data, 1251
  - Data, 1253
  - Data, 1254
  - Data, 1255
  - Data, 1256
  - Data for Display Auxiliary Files, 1242, 1257, 1258, 1259, 1261, 1262, 1263, 1264, 1265, 1266, 1267, 1268
  - Files, 1142, 1145, 1146, 1188
  - Script, 1144, 1146, 1148, 1161, 1167, 1174, 1178, 1180, 1182, 1183, 1186
- Script Mode Introduction, 15
- Search, 486, 507, 534
- Security Constrained OPF, 429, 466, 612, 692, 731, 990, 991, 992, 993, 994, 999, 1002, 1004, 1005, 1009
  - Constraints, 995, 996, 997
  - Controls, 999, 1010
  - Example, 1011, 1014, 1015
  - LP, 1004, 1007, 1008
  - LP OPF, 945, 959, 960, 961
  - OPF Constraints, 938, 939, 940, 942, 943, 950
  - OPF Controls, 937, 948, 952, 964, 980, 981
  - OPF Data, 944, 947, 956
  - OPF Example, 946, 982, 985, 986, 988
  - OPF Options, 947, 948, 950, 952, 956, 959, 960, 961, 962, 963, 964, 965

OPF Records, 944, 966, 968, 969, 972, 973, 974, 976, 978

Select, 32, 297, 298, 312, 313, 317, 320

Select By Criteria, 298, 309

Select OK, 312

Selected group, 299

Selected objects, 261, 298

Selecting Format, 312

Seller, 1017, 1031

Send, 34, 308, 313

    Back, 34, 308, 313

Send All to Excel, 510

Send to Excel, 510

Sensitivities, 754, 855, 861, 863, 867, 871

Sensitivity Calculations, 755

Sequence Data Format, 145

Series Capacitor, 193, 377, 382

    Fields, 382

Series Capacitor Fields, 194

Set Custom Detail, 68

Set Generator Participation Factors, 347, 695

Setting, 303

    Background Color, 303

Setting Default Drawing Options, 254

Shape File Import, 96

Shapefile, 97, 98, 99, 100, 101

    Database Record Dialog, 102

    Export Online as Shapefile, 103

    GIS Shapefile Data Control, 97

    GIS Shapefile Data Format, 100

    GIS Shapefile Data Identify, 98

    GIS Shapefile Data Modify, 99

    GIS Shapefile Data Shape List, 101

Shortcuts, 118

Shortest Path Between Elements, 330

Show Animated Flows, 72

Show Fields Primary, 562

Show Fields Secondary, 564

Show Thumbnail View, 75

Shunts, 157, 184, 195, 249, 250, 276, 286, 361, 411, 445, 608, 833

SimAuto, 1049, 1050, 1051, 1052, 1054, 1055, 1056, 1058, 1061, 1062, 1064, 1065, 1092, 1093, 1111, 1119

    ChangeParametersMultipleElement, 1061

    ChangeParametersMultipleElementFlatInput, 1064

    ChangeParametersSingleElement, 1058

    Delphi, 1073, 1080, 1088

    Functions, 1056, 1057, 1067, 1069, 1071, 1078, 1085, 1086, 1094, 1096, 1098, 1100, 1102, 1104, 1106, 1108

    Get Parameters, 1071, 1073, 1075, 1076, 1078, 1080, 1081, 1082, 1085, 1131

    List of Devices, 1086, 1088, 1089, 1090

    ListOfDevicesAsVariantStrings, 1092

    ListOfDevicesFlatOutput, 1093

    Load State, 1094, 1095

    MATLAB, 1075, 1081, 1089

    Open Case, 1096, 1097

    ProcessAuxFile, 1098, 1099

    Properties, 1111, 1112, 1114, 1116

    RunScriptCommand, 1100, 1101

    Sample Code, 1059, 1062, 1065, 1068, 1070, 1073, 1075, 1076, 1080, 1081, 1082, 1088, 1089, 1090, 1095, 1097, 1099, 1101, 1103, 1105, 1107, 1109, 1113, 1115, 1117

    VB, 1076, 1082, 1090

    Version 9.0, 1121, 1122, 1123, 1124, 1126, 1127, 1128, 1129, 1130, 1131, 1132, 1133, 1134, 1135, 1136, 1137, 1138, 1139, 1140

Simulation, 16, 130

Simulation Control, 23, 49, 56, 688

Simulator Options, 659

Single Linear Step, 1016

Single Solution, 56, 57, 661, 688, 689

Size, 68, 309, 311, 313, 317, 507

    modify, 317

Size/color, 312

Slack, 568

Slack Bus, 568



- Slide Presentations, 82
  - Snap Objects, 301
    - Grid, 301
  - Solution, 16, 584, 622, 659, 660, 661, 667, 669, 671, 673, 674, 675, 677, 679, 684, 685, 691, 693, 697, 699, 700, 701, 702, 703, 704, 705, 708, 709, 734
  - Solution Options, 659, 660, 661, 669, 671, 674, 675, 677, 682, 691, 700, 728, 730, 734
  - Solving, 691
  - Sort, 503, 508
  - Sorting, 499, 503, 508
    - Records, 509
  - Sorting Records, 503
  - Special Data Sections, 1263
  - Specifying and Maintaining a List of Timepoints, 895
  - Spinning Reserves, 568
  - Splitting Buses, 247, 248
  - Stability, 916
  - Stack, 33, 307, 309, 313
  - Stack Level, 33, 307, 309, 313
  - Status Bar, 18
  - Storage, 660, 675
  - Submenu, 66
  - Submode, 15
  - Submodes, 15
    - ATC, 15
    - Case, 15
    - Contingency, 15
    - Fault, 15
    - PowerFlow, 15
    - PV, 15
    - QV, 15
  - Substation, 67, 68, 79, 177, 304, 333, 585, 709
    - Field Options, 333
    - Fields, 177
    - Object, 304
  - Substation Display, 79, 709
  - Substation Display Options, 79
  - Substation Field Options, 333
  - Substation Fields on Onelines, 177
  - Substation Information, 331, 421
  - Substation Records, 585
  - Substation View, 517, 709
  - Substations, 176, 278, 331, 421, 709
  - Super Area, 170, 471, 474, 579
  - Super Area Display Objects, 163
  - Super Area Display Options Dialog, 164
  - Super Area Information, 471, 474
  - Supplemental Classification, 84
  - Supplemental Data, 84
    - Creating Records, 84
    - Fields, 86
    - Linking Display Objects, 85
  - Switched Shunt, 80, 195, 411, 416, 445, 608, 833
  - Switched Shunt Fields, 196, 312, 416
  - Switched Shunt Information, 411, 416, 445, 833
  - Switched Shunts, 196, 249, 250, 276, 411, 416, 445, 451, 567, 572, 833
  - System Scaling, 286
- T**
- Tap, 251, 602
  - Tapping Transmission Lines, 251
  - Taps, 252
  - Terms, 109
    - x-y, 109
  - Text, 68, 216, 217, 312, 313, 507, 534
  - Text Fields, 216
  - Text/font, 309
  - Three Winding Transformers, 409
  - Three-Winding Transformers, 604
  - Throw Over, 782
  - Thumbnail, 67, 75
    - Background, 76
    - Size, 76
  - Thumbnail View, 67, 75

- Tie Lines, 465
- Tie Lines between Areas, 576
- Tie Lines between Zones Display, 578
- Time Step Simulation, 873, 874, 876, 878, 879, 880, 883, 884, 885, 888, 889, 890, 891, 892, 895, 896, 897, 898, 899, 901, 902, 903, 905, 906, 908, 909, 910, 913
  - Binding Constraint Dialog, 888
  - Change Timepoint Time Dialog, 897
  - Custom Results Selection Dialog, 890
  - Dialog, 876
  - Hourly Summary, 880
  - Input Page, 883
  - Insert/Scale Column Dialog, 899
  - Loading Hourly Input Data, 901
  - Matrix Grids, 884
  - New Timepoint Dialog, 896
  - Options, 892
  - Pages, 879
  - Quick Start, 874
  - Results, 889
  - Results Grid Pages, 891
  - Results: Constraints, 885
  - Running a Timed Simulation, 909
  - Running OPF and SCOPF, 910
  - Schedule Dialog, 903
  - Schedule Subscription Dialog, 906
  - Schedule Subscriptions Page, 908
  - Schedules Page, 905
  - Setting up Scheduled Input Data, 902
  - Specifying and Maintaining a List of Timepoints, 895
  - Storing Input Data and Results, 913
  - Toolbar, 878
  - TSB Case Description Page, 898
- TLR, 866, 867, 869, 871
- TLR Multiple Device Type, 869
- TLR sensitivities, 866, 867
- TLR Sensitivities, 871
- Toggle, 65
- Toggle Full Screen, 66
- Tools, 303, 306
- Top, 34, 308, 313
  - level, 313
- Topological Differences, 726
- Total Digits, 311
- Transactions, 573, 575, 621, 622, 697, 704
- Transfer, 1033, 1039
- Transfer Limit, 1039
- Transfer Limit For Scenario, 1033
- Transfer Limiters, 1033, 1039
- Transfer Limiters Display, 1039
- Transfer Result Reporting Options, 1026
- Transformer AVR Dialog, 375, 401, 438
- Transformer Control, 373, 397
- Transformer Fields, 192, 312
- Transformer Impedance Correction, 400
- Transformer Impedance Correction Tables, 444
- Transformer Options, 357
- Transformer Phase Shifting, 405, 442
- Transformers, 157, 184, 190, 192, 249, 250, 357, 361, 373, 375, 383, 397, 399, 400, 401, 403, 405, 407, 409, 434, 438, 440, 442, 444, 567, 598, 600, 602, 604, 629, 637, 685, 829
  - Control, 373, 397
  - Fields, 383, 407
- Transmission Line, 389, 390
  - DC Actual Flow, 390
  - DC Inverter Parameters, 389
- Transmission Line Information, 829
- Transmission Line Options, 357
- Transmission Line Parameter Calculator, 363
- Transmission Line Transformer Options Display, 359
- Transmission Line/Transformer Options, 360, 373, 377, 397
  - Parameters/Display, 360
  - Series Capacitor, 377
  - Transformer Control, 373, 397

Transmission Lines, 182, 183, 188, 189, 190, 249, 250, 272, 357, 361, 380, 385, 386, 388, 434, 567, 598, 600, 606, 629, 637, 685, 829

DC, 385

DC Line Parameters, 386

DC Rectifier Parameters, 388

Fields, 380

Transmission Loading Relief, 866, 867, 869, 871

Transmission Loading Relief Sensitivities, 866, 867, 871

Turns Ratio, 400

Tutorial, 12, 807, 808, 1329

Contingency Analysis - Page 10, 821

Contingency Analysis - Page 11, 822

Contingency Analysis - Page 4, 813

Contingency Analysis - Page 5, 815

Contingency Analysis - Page 6, 816

Contingency Analysis - Page 7, 818

Contingency Analysis - Page 8, 819

Contingency Analysis - Page 9, 820

Tutorial Contingency Analysis, 807, 1329

Tutorial Creating a New Case, 1278

Tutorial Solving an OPF, 1322

Tutorial Starting with an Existing Case, 1307

Tutorial Contingency Analysis, 808

Tutorials, 807, 1278, 1279, 1281, 1283, 1284, 1287, 1289, 1291, 1293, 1296, 1298, 1302, 1304, 1307, 1308, 1309, 1310, 1311, 1312, 1313, 1314, 1315, 1316, 1317, 1318, 1319, 1320, 1321, 1322, 1323, 1324, 1325, 1326, 1327, 1329

## U

Uncheck, 317

Anchored, 317

Undo, 321

Unit Type, 592

Unlinked Display Objects, 259

Unlinked Objects, 259, 301

Highlighting, 301

Unused Bus Numbers, 265

Use, 317

Display/Size Tab, 317

Use Absolute Values, 68

Mvar, 68

MW, 68

Use Area/Zone/Owner Filters, 26, 508, 511

Use Background Fill, 312

Use Conditional Display, 315

Using Difference Flows, 724

Using the Online Alignment Grid, 302

## V

Validate, 16

Variable O&M, 591

Version 11.0, 5

Version 12.0, 5

View, 46, 67, 75, 112, 156, 204, 706

View Name, 112

Violations, 771

Visualize, 848, 850

Power Flows, 848, 850

PTDFs, 848

Visualizing, 69, 679

Out-of-Service Elements, 69

Voltage Adequacy, 916

Voltage Control, 425

Voltage Gauges, 174, 175

Voltage Sensitivities, 861

Voltage Stability, 916

## W

Warning, 70

Warning Color, 70

Warning Percent, 70

Default values, 70

Warning Size Scalar field, 70

Web, 9

Web Publishing, 148, 505

Weighted Transmission Loading Relief, 870

Why is this field or line disabled, 499

Window, 11, 16, 509

Windows Basics, 11

Word, 82

Word Processor, 82

## X

X Grid Spacing, 301

XF, 602

x-y, 109, 112

## Y

Y Grid Spacing, 301

Ybus, 149, 639

## Z

Zero Sequence, 830

Zone, 168, 298, 304, 323, 345, 430, 449, 521

Object, 304

Zone Display Objects, 161

Zone Display Options Dialog, 162

Zone Fields, 168

Zone Information, 323, 449

Zones, 451, 521, 568, 610

Zoom, 33, 46, 68, 70, 75, 110, 298, 301, 304, 307, 313, 314, 315

Area, 46

In, 46

Levels, 46, 70, 75, 110, 112, 298, 301, 313, 314, 315

Out, 46

Pan, 110

Percentage, 70

Toolbar, 46

Zoom/Pan/Find, 110, 304, 313